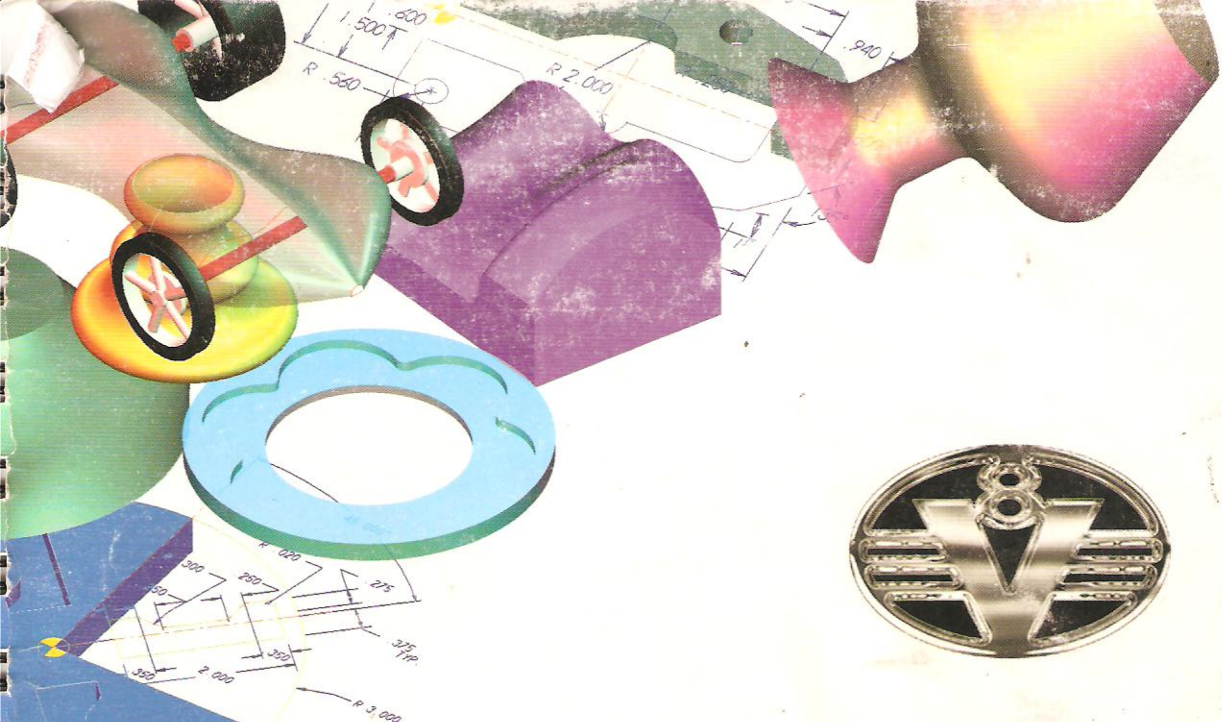
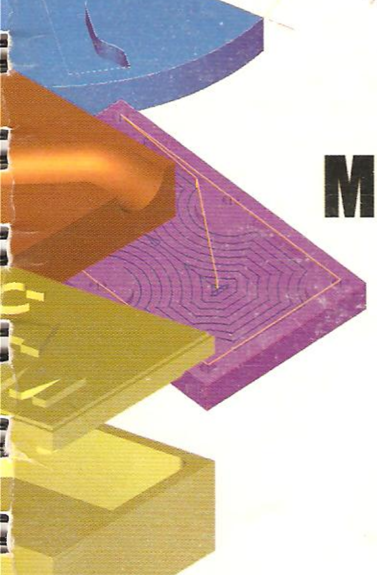




Learning Series
Mill Projects & Applications

Mastercam®

When Second Best Won't Cut It.

Mastercam®

V8

Learning Series

Mill Projects and Applications

Mastercam Version 8

Table of Contents

Introduction

About using and learning Mastercam

Welcome to Mastercam Version 8. Mastercam is a powerful CAD/CAM application that lets you design parts and choose from 2- through 5-axis milling, turning, wire EDM, lasers, mold base development, surface, and solid modeling.

To help you learn Mastercam, extensive online help and this manual accompany the product.

About this book

This guide teaches you the practical and efficient use of Mastercam Design and Mill software. To get you started, the guide provides a short tutorial on learning the product user interface and then launches into many practical projects and tutorials for you to complete. The projects and tutorials are designed to show you efficient ways to design and mill real-world parts.

Note: You must have Level 1 of Mastercam Mill Version 8 or a higher level installed on your PC to complete Chapters 2, 3, 4, 5, 6, 8, 9, 10, 11, 12 and 18 and Level 3 to complete Chapters 7, 13, 14, 15, 16, and 17.

Typographic conventions used

This guide uses several typographic conventions:

Bold type identifies any portion of the Mastercam interface that you select, including menu options, dialog box options, and buttons. (Example: the **MAIN MENU**)

Brackets [ ] identify keys that you press on the keyboard, such as [Enter], [Esc], etc.

Italic text identifies messages that display in the prompt area of the Mastercam main screen where you enter values. (Example: *Enter coordinates*)

Note: A point of information related to the preceding text.

The [Esc] key is normally used throughout Mastercam to back up to the previous level of the menu structure. When the keyboard convention does not apply, the use of the [Esc] key is explained in the text.

As shown in the following example, check boxes adjacent to buttons on a dialog box activate or deactivate the button. If the check box is selected, the button is active and can be chosen to edit the related options. If the check box is not selected, the button cannot be chosen and all features associated with the button are disabled.

Online help

Online help contains the latest and most up-to-date information about Mastercam. To access online help, press the Help button on a dialog box or the toolbar, or press [Alt+H] at any menu while you are working in Mastercam.

1 Getting Started with Mastercam

Exercise 1 – Learning the Mastercam interface

This exercise shows you how to navigate through Mastercam. You will:

- **Launch the application**
- **Make selections from the menus and toolbar**
- **Use program prompts for guidance**

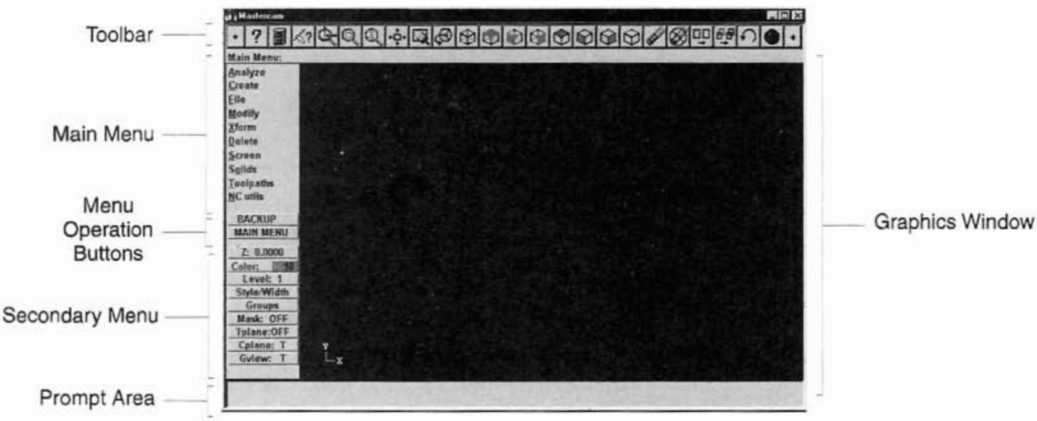
▶ Launch Mastercam

You must have Level 1 of Mastercam Mill Version 8 or a higher level installed on your PC to complete Chapters 2, 3, 4, 5, 6, 8, 9, 10, 11, 12 and 18 and Level 3 to complete Chapters 7, 13, 14, 15, 16, and 17.

1. Choose
   - **Windows Start menu**
   - **Programs, Mastercam, Mill 8**

Note: If you cannot open Mastercam, refer to your installation instructions included in a separate document.

2. The main Mastercam window appears, as shown below.



▶ *Use the Main Menu*

The Main Menu contains the primary functions you use in Mastercam.

1. Choose
   - **MAIN MENU**
   - **Create**
   - **Point**
   - **Position**
2. Notice that the prompt area at the bottom left of the screen displays the message *Create point: specify a point.*

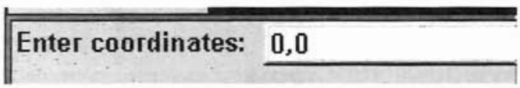
▶ *Use the prompt area*

The prompt area displays a message to help you understand what action to take. It is also the screen location where you enter values.

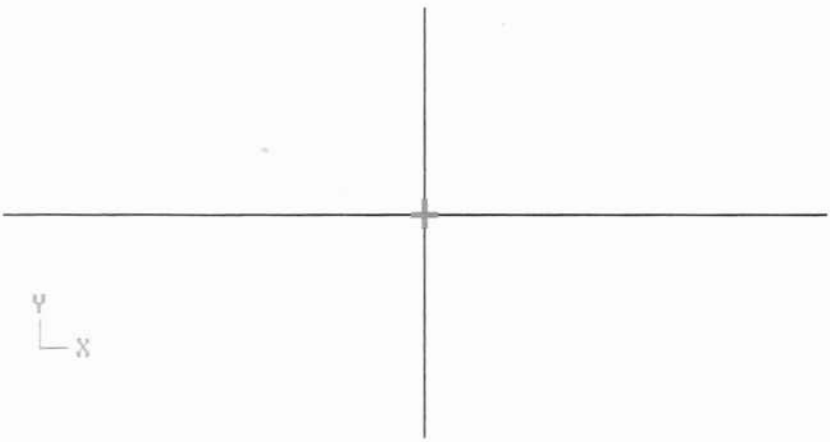
1. Without moving your mouse, type **0,0**.

Note: Be sure to enter the number zero and not the letter "O."

2. Notice that the values you type (the coordinates) appear in the prompt area as you type.



3. Press [Enter] to display the point at position 0,0.
4. Press [F9] to display the construction origin. The graphics window should look like the following picture.



Exercise 2 – Designing a rectangle

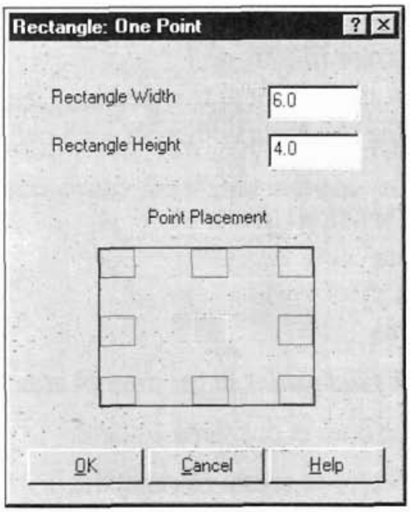
This exercise shows you how to design a simple rectangle with fillets. You will:

- **Use the Create, Rectangle function**
- **Use the Create, Fillet function**

Create a rectangle

In this task, you will create a rectangle with the center at 0,0.

1. Choose
   - **MAIN MENU**
   - **Create**
   - **Rectangle**
   - **1 Point**
2. Enter the values shown on the dialog.

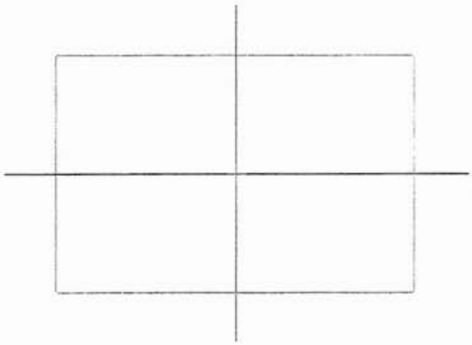


3. Choose **OK**.
4. Drag the cross-hairs cursor near to the point you created earlier until you see a small white square appear over the point.

5. Click the mouse button to position the rectangle so that its center is at the coordinates 0,0. Notice that the rectangle function is still active (in case you want to create another rectangle).
6. Press [Esc] to exit the rectangle function.

Note: If you accidentally create more than one rectangle, you can delete the unwanted geometry by choosing MAIN MENU, Delete and selecting the lines to delete.

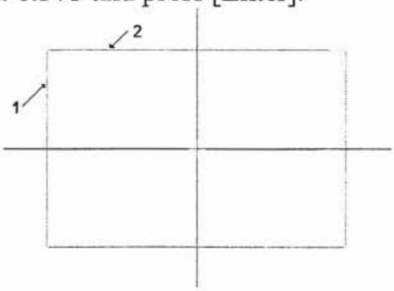
Your part should look like the following picture.



Create corner fillets

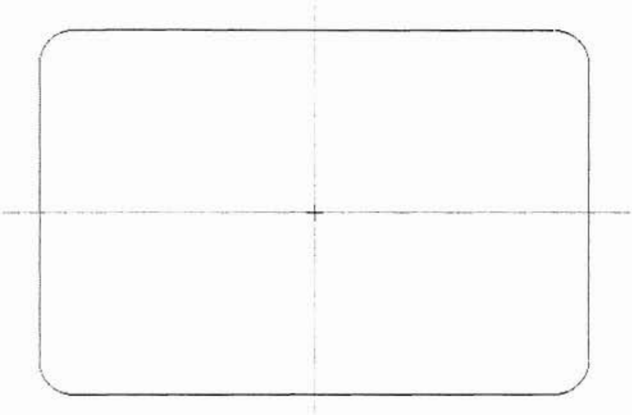
A fillet is an arc tangent to two entities. Create fillets to make rounded corners on the rectangle with a 3/8 inch radius.

1. Choose
   - **MAIN MENU**
   - **Create**
   - **Fillet**
   - **Radius**
2. *Enter the fillet radius* in the prompt area: **0.375** and press [Enter].
3. Select the lines at positions 1 and 2.
4. Repeat Step 3 to create fillets at the other three corners.



Tip: When you move the cursor close to an entity during selection, it highlights in white.

Your part should look like the following picture.

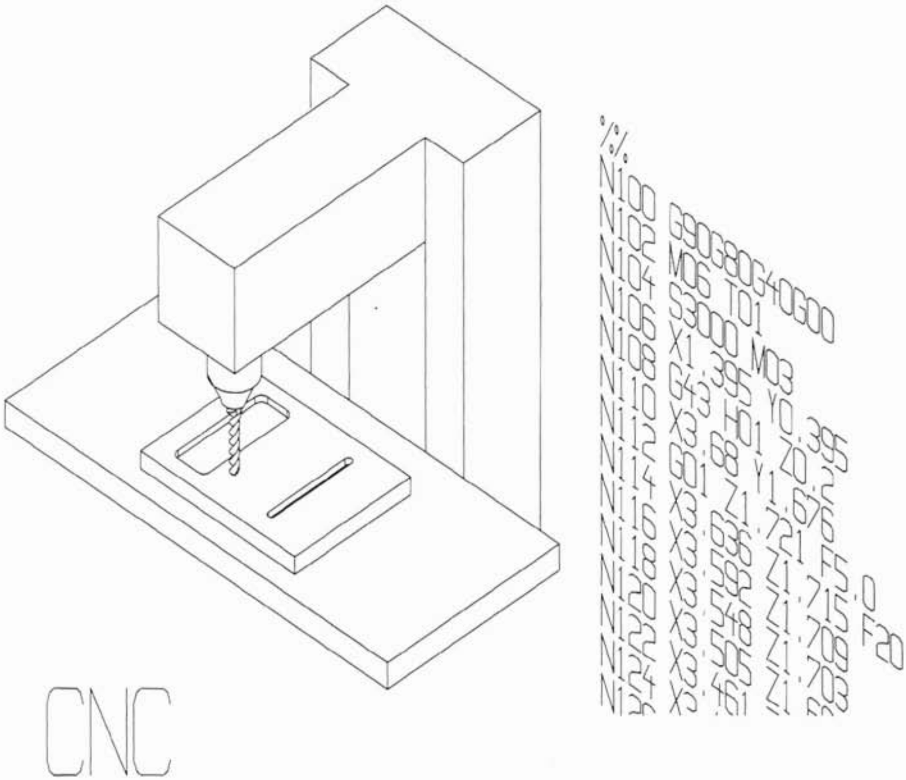


Save the file

1. Check with your instructor on how and where to save your files.
2. Choose
   - **MAIN MENU**
   - **File**
   - **Save**

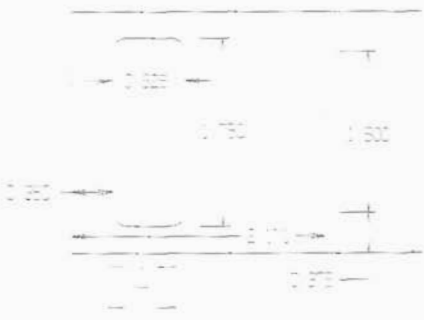
Note: It is a good idea to save your file frequently as you work. This way, if you make an error, you can choose File, Get to open a previously saved version of the file. Each time you successfully finish one exercise, save your file.

CAD/CAM/CNC Overview

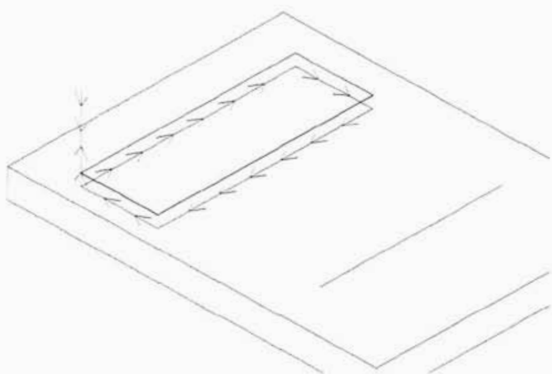


Definitions

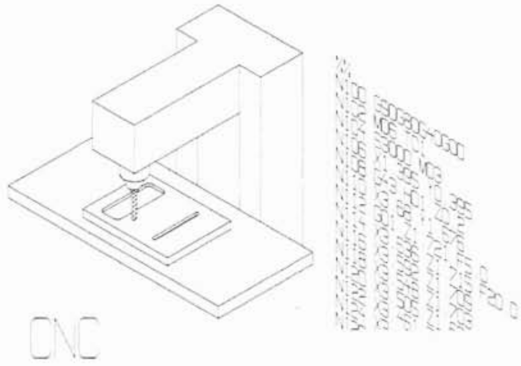
CAD (Computer Aided Drafting) - Mechanical drafting performed on a computer to create part geometry.



CAM (Computer Aided Machining) - Produces toolpaths from CAD geometry and creates machine code for the CNC.



CNC (Computer Numerical Control) - CNC is a computer technology for controlling machine tools, including milling machines. The CNC computer, referred to as a controller, feeds numerical data to the motors. The motors physically move the table to cut the stock. CNC technology is also used on lathes, wire Electro-Discharge-Machines (EDM), sheet metal punches and brakes, cutting torches, lasers and more.



CAD

To start the process, a CAD drawing of the intended part is needed. Figure 1 is an example of a part drawn in a CAD system. This is commonly referred to as geometry.

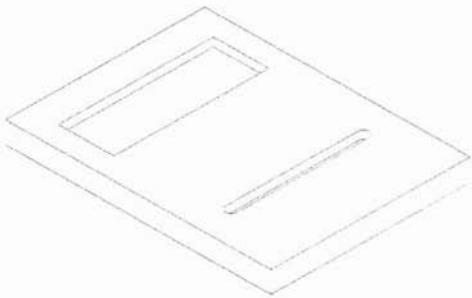


Figure 1. CAD Geometry with Hidden Lines Removed

When naming the geometry, Mastercam automatically adds the file extension of MC8. If the part were called "PART1", it would be saved on the disk as PART1.MC8.

Notice in Figure 1, there are no dimensions. Dimensions are not necessary when programming with CAM, as the geometry itself defines the part. After the geometry is created, the next step is to tell the computer which lines are used for each machining process.

CAM

Lines are linked together to form a continuous boundary around the area to be machined. This identifies which lines will be used for the toolpath. Linking the lines together is referred to as chaining. Figure 2 shows the geometry . Only the inner rectangle is chained. All other lines are not used at this time. Only the lines that are chained will be used to calculate the toolpath.

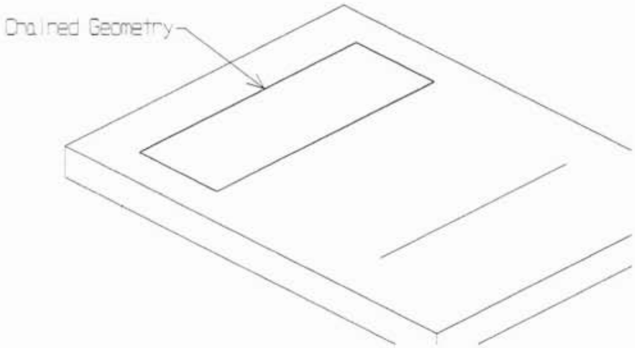


Figure 2. Part with Geometry Chained

Once the chaining is complete, the computer needs some information on the size and shape of the cutter, how fast to cut, spindle speed, etc. All of this data is handled in the parameter screens. After this information is in the program, the computer can generate a toolpath.

An example of the toolpath is seen in Figure 3.

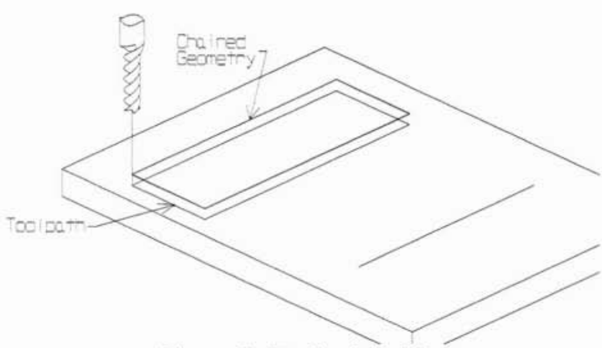


Figure 3. Toolpath in Mastercam

The toolpath is a sequence of lines and arcs that describe the tool position.

Figure 4 is the same toolpath with arrows showing the direction of each cut as it makes its way around the part.

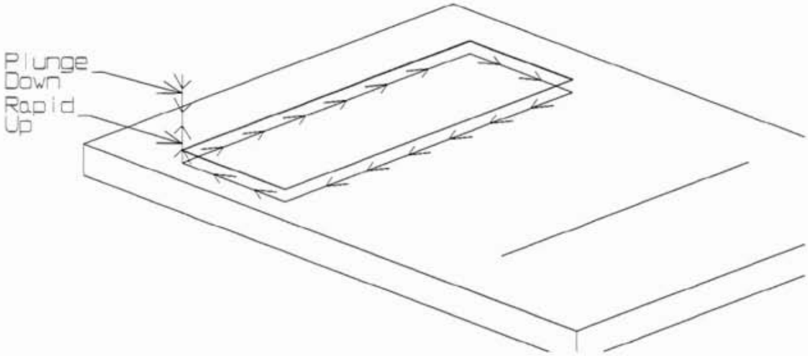


Figure 4. Contour Toolpath as a Continuous Line

Notice that the plunge lines and rapid line in Figure 4 are in the same location. The tool starts above the part and plunges to the specified depth of the contour. The endmill then cuts around the chained geometry and moves up out of the part at the same location that it plunged into the stock. The retraction out of the part is called a rapid move. A rapid move is machine movement at the maximum speed.

In Mastercam, the toolpath file has an extension called NCI. The toolpath is commonly called an NCI file. To follow with the previous example, the NCI file would be PART1.NCI.

It may be prohibitive to machine the entire part in one operation because it is too deep to cut with one pass. To avoid damage to the cutter or equipment, it may be necessary to machine the shallow depths first and then return to the deepest areas.

The process of milling usually requires several steps and tools. If another toolpath were desired, such as machining the slot on the right side of the part, it could be done next. This example, however, has only one toolpath.

Before the milling machine can use the toolpath information in the NCI file, it must be converted to the same language as the CNC machine.

In Mastercam, this translation is referred to as post-processing, or posting. Post processors are specific to the machine since there are many different CNC languages. The program has to match the language exactly.

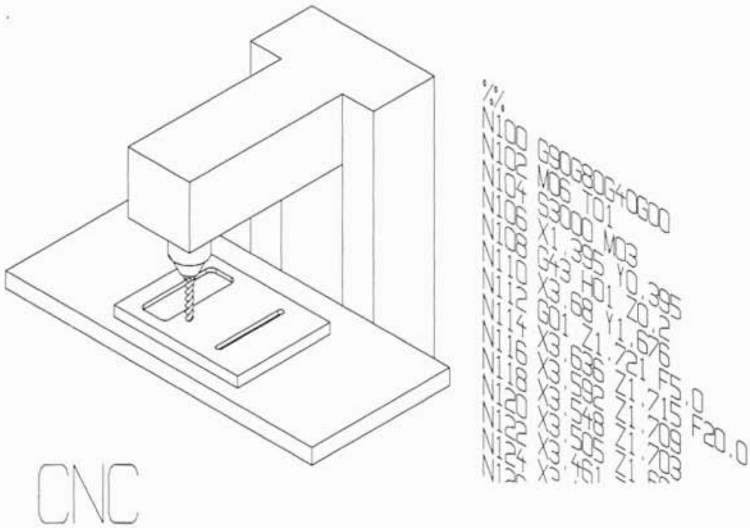
The post processor reads the information stored in the NCI file and writes a new file with the completed CNC code. Post-processing does not change the NCI file. The CNC code file extension is called NC. The file name in this example would be PART1.NC.

The NCI file seen in Figures 3 and 4 was post processed for three different CNC machines, DYNA, LIGHT MACHINES, and the industry standard, FANUC. Notice the similarities and differences between the following codes.

DYNA MECHTRONICS	FANUC	LIGHT MACHINES
001 START INS 01	%%	N001G90G00M03
002 FR XY = 10.	N001G90G80G40G00	N002Z.6
003 SETUP>ZCXYU	N002M06T1H1	N003G0X.85Y.375
004 SPINDLE ON S2000	N003M03	N004G1Z-.1F5.
005 FR XYZ = 30.	N004G0X.85Y.375	N005Y2.125F10.
006 GO X .85	N005G43H1Z.6	N006X1.475
007 GO Y .375	N006G1Z-.1F5.	N007Y.375
008 GO Z .6	N007Y2.125F10.	N008X.85
009 FR Z = 5.	N008X1.475	N009G0Z.6
010 GO Z -.1	N009Y.375	N010M05
011 FR XY = 10.	N010X.85	N011X0.Y0.
012 GO Y 2.125	N011G0Z.6	N012M02
013 GO X 1.475	N012G28M05	
014 GO Y .375	N013M30	
015 GO X .85	%	
016 FR XYZ = 30.		
017 GO Z .6		
018 SPINDLE OFF		
019 END NEWPART		

To review:
A geometry file with *******.MC8
A toolpath file with *******.NCI
A CNC code file with *******.NC
have all been made in that order. The NC file is now ready to be sent to the milling machine controller.

CNC



The device that controls the CNC milling machine is called the controller. It analyzes the machine code and converts that information into electrical signals that coordinate the motors to drive each movement on the CNC machine.

Remember, the machine code contains statements telling the milling machine to go from one position to another.

If the NC file had a statement that read Z-.5, the controller would send electrical signals to the motor on the Z axis until the cutter was one-half inch lower than when it started.

As the machine performs each move, the finished part begins to take shape, until the part is finally finished.

CNC Machining

This overview will help you understand CNC before using the machinery.

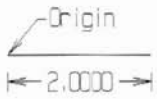
CNC stands for Computer Numerical Control. Automated machines use motors to drive each axis. A computer controls the direction and length of time each motor rotates.

The computer that controls the milling machine is called the controller. The controller also synchronizes the motors to produce arcs that would be extremely difficult to produce by hand.

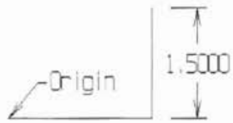
Program

Each shape in Mastercam is translated into XYZ moves, similar to plotting points on a graph. Remember that X moves are horizontal and Y moves are vertical. The coordinate moves for a simple rectangle would look like the following example:

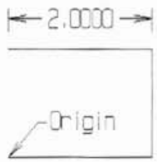
From X0,Y0 go X2 inches



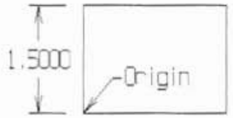
Go Y1.5 inches



Go X-2 inches



Go Y-1.5 inches



The rectangle is complete.

Add tool down and tool up movements to show when the tool is above the stock (Z positive) or in the stock (Z negative). The coordinates would look like this:

```
Z-.2
X2
Y1.5
X-2
Y-1.5
Z.2
```

Post processing

Post processing converts the coordinates to the CNC machine language. The program could look like the example at right with codes necessary for the controller to carry out the program.

```
%%
N001G91G40G00
N002M06T1H1
N003M03
N004G0X0.Y0.
N005G1Z-.2F5.
N006X2.F10.
N007Y1.5
N008X-2.
N009Y-1.5.
N010G0Z.2
```

This is Fanuc-based CNC language for the rectangle. Notice the similarities between the last two examples: the first has the same information as this one, with several additions:

Percent signs are needed to transfer the program into and out of the controller.

Line numbers are added to each line to make it easy to find specific locations, as demonstrated in the following items:

Line N001 has "G" codes that set the controller to incremental moves, no cutter compensation and move at rapid travel.

Line N002 is a tool change for tool number 1 with offset length number equal to 1.

Line N003 is spindle on.

Line N004 moves cutter to program reference zero (PRZ).

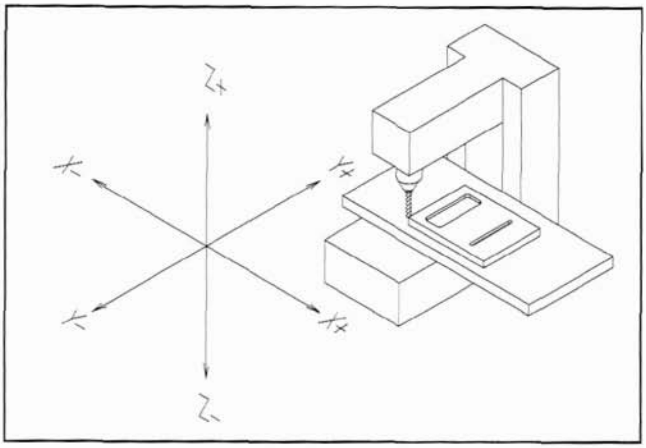
Line N005 has a slow plunge rate of 5"/min. and lowers the tool into the stock.

Line N006 has a slow feed rate of 10"/min.

Line N010 retracts the tool at the rapid feed rate out of the stock.

Line N011 is the end of the program.

Relating these movements to the milling machine, the axes are in the orientation shown in the graphic at right. X positive is to the right, Y positive is towards the back, and Z positive is up.



Note: Do not confuse the tool movement with the table movement. For the tool to move X 2 inches, the table has to move to the left 2 inches.

In other words, for the tool to move to the right on the part, the table has to go left.

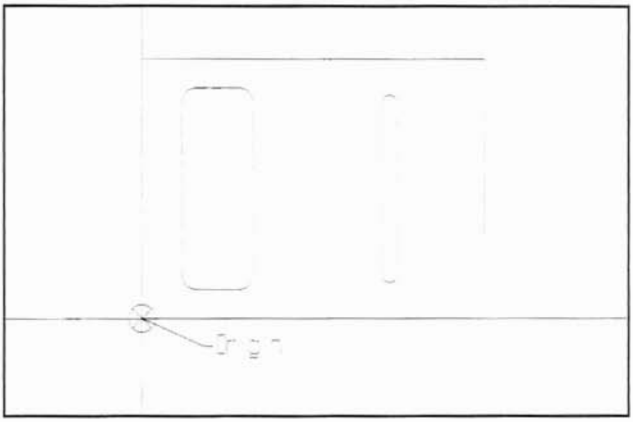
Regardless of the type of machine, a successful program involves certain steps. One of them is setting the Program Reference Zero (PRZ).

Setting the PRZ

After the program is loaded into the mill, the mill must be jogged to the PRZ and then set.

The PRZ must be in the same location as the origin in the CAM software. Usually the PRZ is in the lower left-hand corner of the part at the top of the stock.

The following picture is a part with the origin in the lower left corner. The following graphics are setting the PRZ on the mill at the top of the stock.

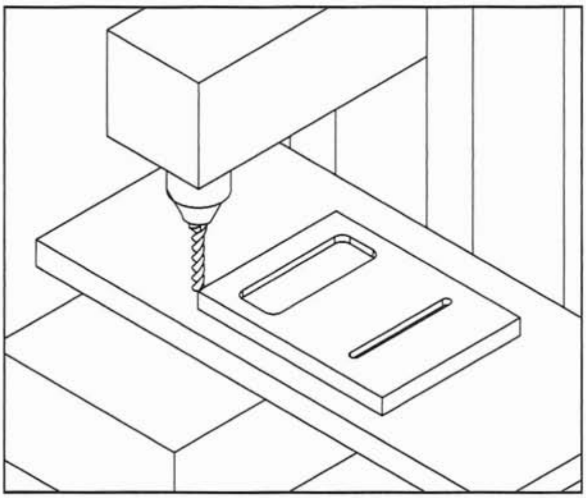


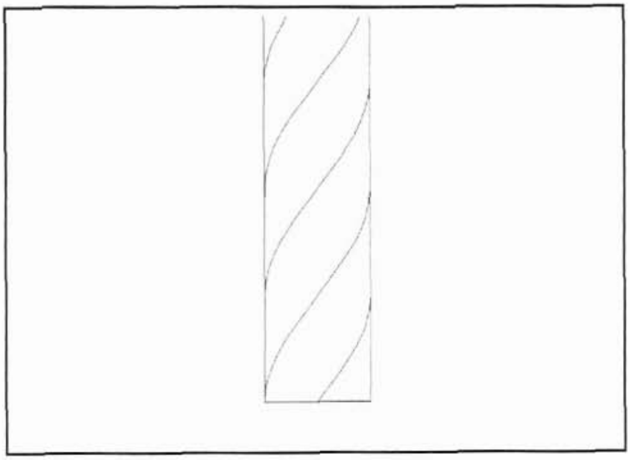
There are advantages to having the PRZ in this location.

It allows all geometry creation to be made in the positive X, positive Y quadrant on a CAD/CAM system.

The corner of the stock is relatively easy to find.

Z depth values are somewhat easier to calculate. All negative values are below the surface of the stock.





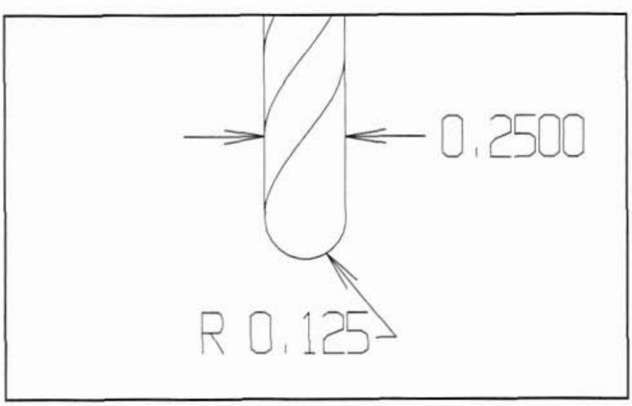
Endmills

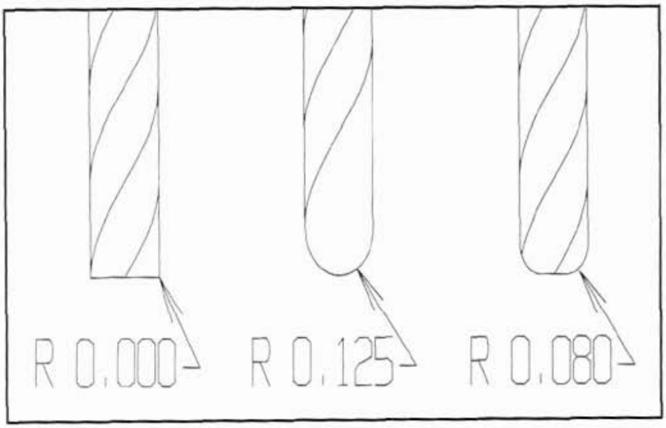
Endmills come in different shapes and sizes, each with its own purpose.

Flat endmills, like the one shown in the preceding picture, are good for facing the top of the stock. It makes the top of the stock flat because the bottom of the cutter is flat.

Some flat endmills are made to cut on the side of the cutter only. This means the tool cannot plunge straight down into the stock as a drill bit. Flat endmills may have to ramp down into the stock at an angle to reach the proper depth.

A ball endmill has a hemispherical (round) bottom, shown in the following graphic. Ball endmills produce shaped surfaces.





Cutter diameter is the distance from one side of the endmill to the other.

The corner radius is the radius between the side of the cutter and the bottom of the cutter, looking at it from the side.

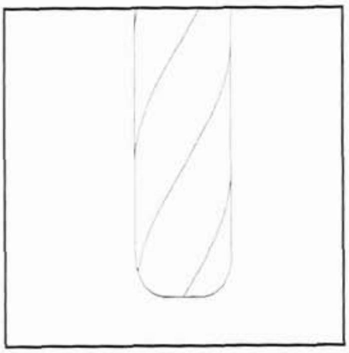
The preceding graphic shows three endmills from the side view. The corner radius can be seen from this view.

A flat endmill has a corner radius of 0.0". There is no radius between the flat bottom and the side.

A ball endmill has a corner radius equal to half the cutter diameter. A 0.250 inch cutter has a corner radius of 0.125".

If a cutter has a corner radius less than the radius of the cutter, it is said to be a bull nose. A close-up example of this can be seen in the following graphic.

Bull nose cutters are flat on the bottom but have a corner radius, combining the benefits of flat and ball endmills.

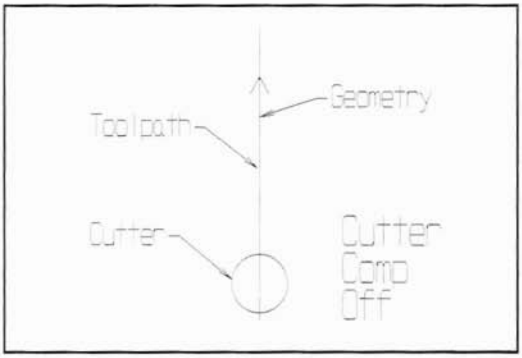


Cutter control

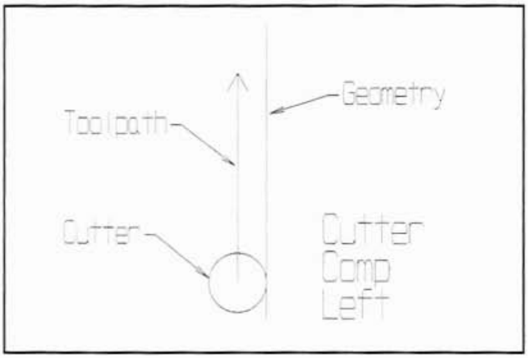
Controlling the cutter means choosing a toolpath for the cutter in relation to the geometry. Two aspects of cutter control discussed in this section are cutter compensation and climb vs. conventional milling.

The first example shows no cutter compensation. In Mastercam, this is called cutter comp-off. The first shows a cutter with no compensation and a contour from the top view.

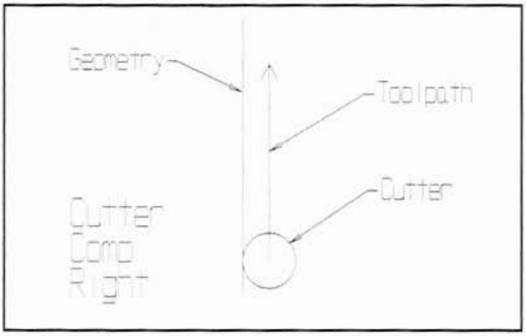
With the cutter comp off, the endmill cuts directly on top of the line. Half of the diameter is on each side of the line.



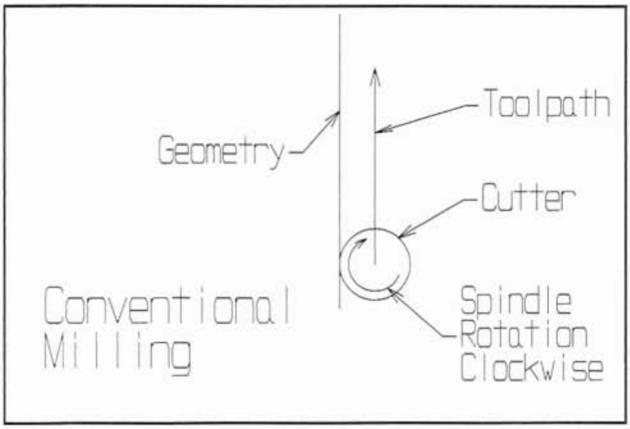
Cutter comp-left: The cutter travels along the line with the cutter on the left side of the line or geometry. The entire diameter of the cutter is on the left side of the line, as shown at right.



Cutter comp-right: In the example at right, the cutter is on the right side of the line with the edge touching the line .

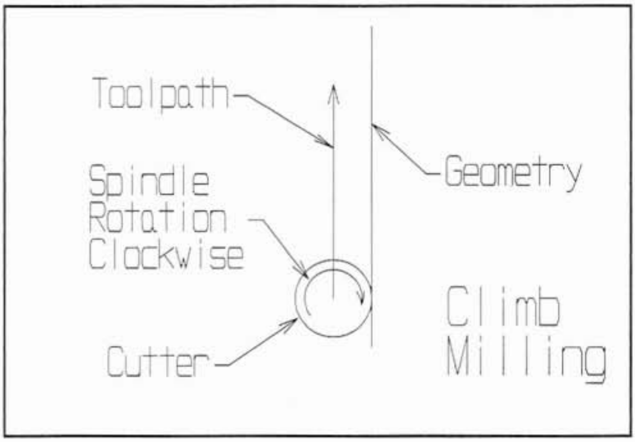


Different finishes can be produced on the part depending on the direction the cutter travels around the part. The type of milling is determined by the cutter compensation and direction of the tool.



At right is an example of conventional milling. At the point where the cutting surface of the tool contacts the part, the edge of the cutter is traveling in the same direction as the tool.

The next graphic is an example of climb milling. The contact surface of the cutter is traveling in the opposite direction of the cutter. The cutter tends to climb along the part in the direction of cutter travel.



Conventional milling is useful for hogging material; climb milling is useful for producing smooth finish passes. Hogging refers to gross milling where a great deal of material is scooped out of the part.

Feed rates and spindle speeds

Each material varies in characteristics: some soft (wax), some harder (aluminum), and others harder yet (stainless steel).

Because characteristics vary, it is important to select the appropriate cutter, spindle speed and feed rate for each material. Failure to do so can overheat and dull an endmill, or even break it. This can damage the stock, and possibly the CNC machine.

Diameter of the cutter, number of teeth, chip load, and surface feet per minute all play an important part in determining which cutter to use and what feed rate / spindle speed to use for a particular material.

For comparison, the following chart has data for milling stainless steel and aluminum using a 1/4" cutter with 2 flutes:

Material	Cutter Dia. /	Teeth /	Surface speed /	Chip load /	RPM /	Feed rate
Aluminum	.25"	2	300 fpm	.005	4584	45.0 ipm
Stainless	.25"	2	35 fpm	.003	535	3.2 ipm

Notice that aluminum has a much larger surface speed than stainless steel and a slightly larger chip load per tooth. Both of these values result in a higher spindle speed and feed rate.

Refer to the *Machinery's Handbook* to find the specific feeds and speeds for a particular material and cutter. Non-machinists can also find a wealth of information in this book.

Important tips

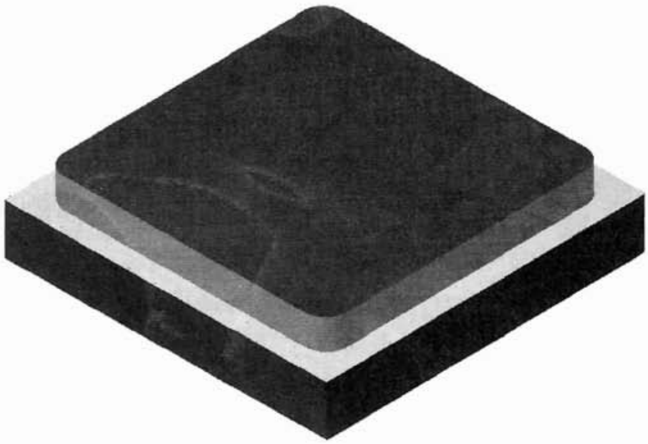
Safety

- Never operate any equipment without proper permission.
- Always wear safety glasses.
- Pay attention to the mill at all times during operation. While this may seem obvious, it sometimes helps to be reminded. This includes **watching** and **listening** for problems.
- Keep a hand near the emergency shut-off button. While it may be embarrassing to shut the machine off, it will be worse if something breaks.

Feeds and speeds

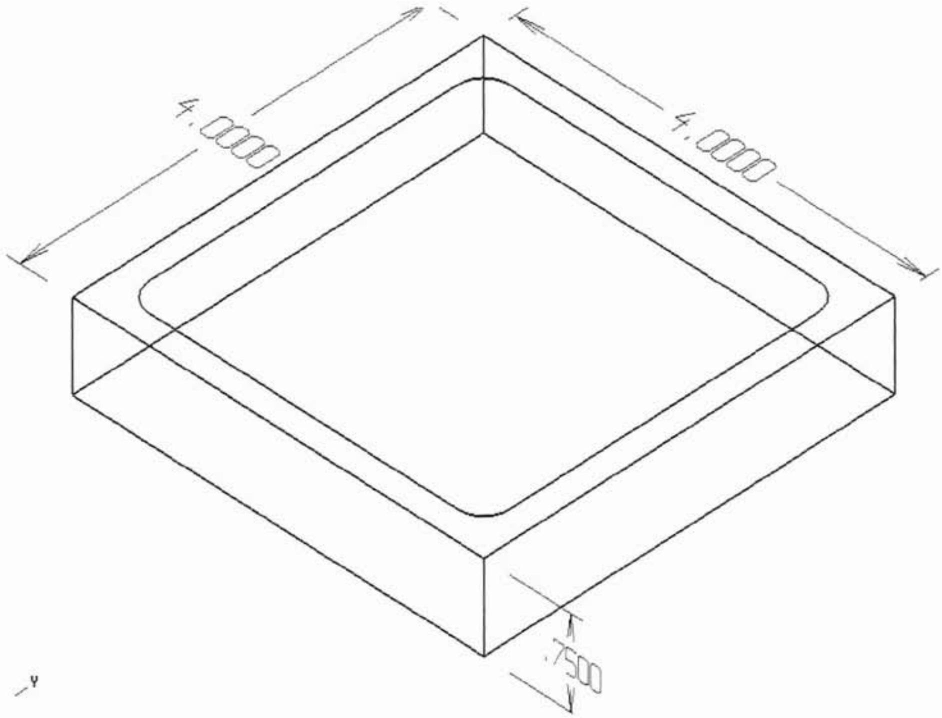
- Start with non-ferrous materials such as wax, wood, or plastic. They are much more forgiving to improper feeds and speeds. Endmills could last much longer using these soft materials.
- Never plunge more than 50% of the diameter of the endmill into the stock. For example, a .5 inch endmill should not cut deeper than .25 inches per Z depth pass.
- Each pass of the cutter should not move over more than 50% of the cutter diameter when pocketing (removing material from an area). Doing this may not produce the most efficient toolpath but will save cutters until a feel for the material is acquired.
- Set the plunge rate at half of the feed rate. The highest stress on the cutter is in the plunge mode, so reducing the feed rate while plunging will save many cutters.

2 Jewel Box Lid - Bottom



Objectives

- Navigate the Mastercam menus.
- Create lines and arcs.
- Translate and offset lines.
- Enter coordinates.
- Construct 3D geometry.
- Chain the geometry.
- Create a contour toolpath.
- Enter parameters for a contour toolpath.
- Perform solid model verification of the toolpath.



Geometry creation

▶ *Create a rectangle*

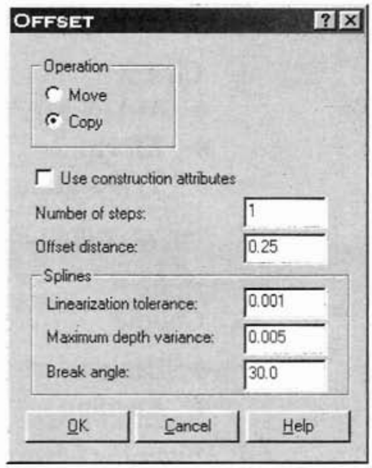
1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Rectangle**
   - ◆ **2 points**
2. *Enter the coordinates.* **X0,Y0**

Note: Be sure to enter the number zero and not the letter "O".

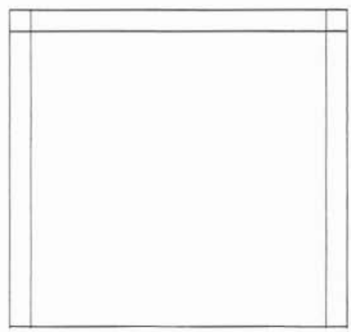
3. Press [Enter].
4. *Enter the coordinates.* **X4, Y4**
5. Press [Enter].

6. Choose **Fit** on the toolbar to center the geometry.

▶ *Create the inside entities*



1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Xform**
   - ◆ **Offset**
2. Enter the values as shown in the dialog box at right, being sure to select Copy.
3. Choose **OK**.
4. Select a line to offset. Select any one of the lines of the rectangle by clicking on it with your mouse.
5. Indicate the offset position by clicking anywhere inside the rectangle.
6. Repeat the same process for the three remaining lines. The drawing should now look like the drawing at right.



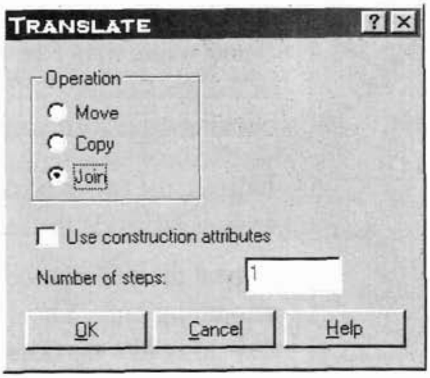
▶ *Fillet the corners*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Fillet**
   - ◆ **Radius**
2. *Enter the fillet radius.* **0.25**
3. *Select an entity.* Click on one of the two new lines as shown in the diagram at right.
4. *Select another entity.* Click on the other new line as shown.
5. Repeat the above step for the remaining three corners.

▶ *Make the drawing 3D*

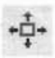
1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Xform**
   - ◆ **Translate**
2. Select entities to translate by clicking on the four outside lines of the rectangle.
3. Choose
   - ◆ **Done**
   - ◆ **Rectang**
4. *Enter the translation vector.*

   Z-.75
5. Enter the values shown on the dialog box at right.



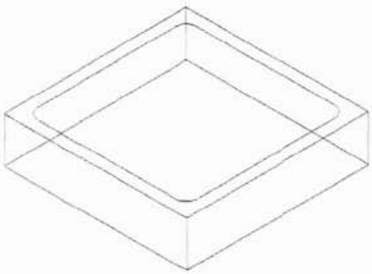
6. Choose **OK**.

7. Choose the **Gview (isometric)** button from the toolbar to change the graphics view.



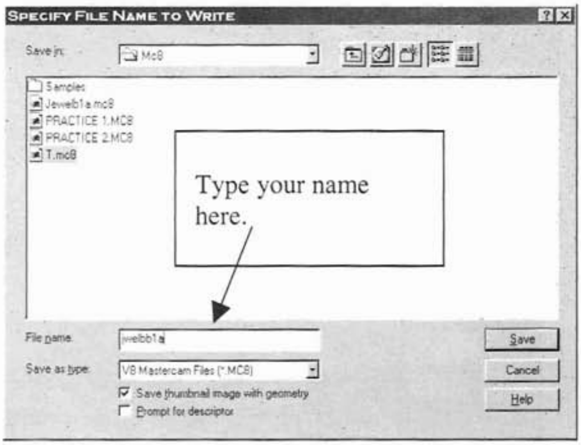
8. Choose the **Fit** button from the toolbar to fit the geometry in the graphics window.

The part should look like the following picture.



▶ *Save the drawing*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **File**
   - ◆ **Save**
2. Save by typing the first 6 letters of your last name, followed by 2 (eg: johnso2) in the file name space as shown at right. Mastercam will add the extension automatically.

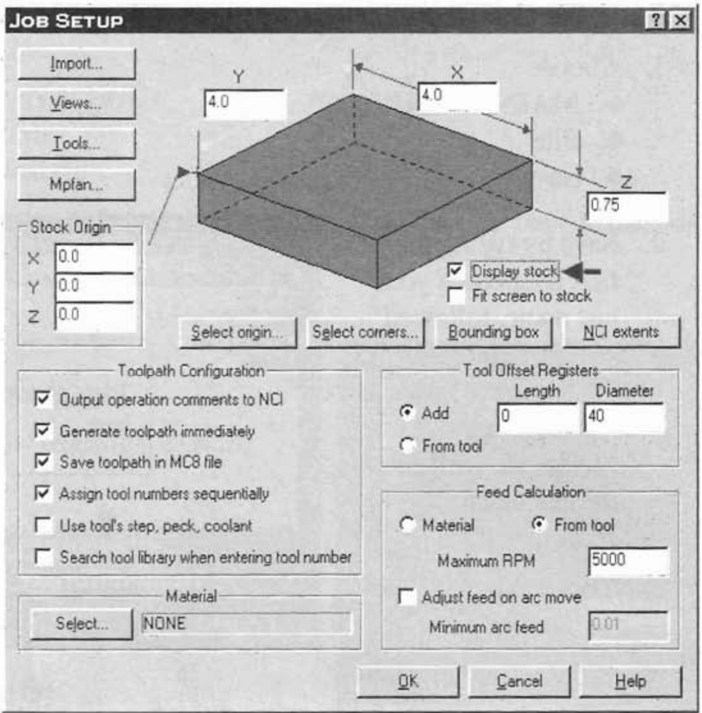


3. Choose **Save**.

Toolpath creation

▶ *Define the rough stock*

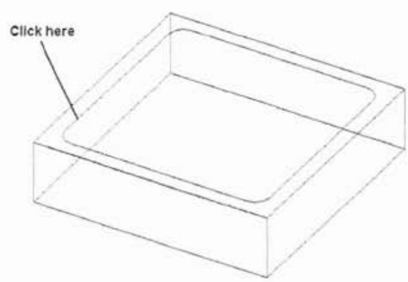
1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Toolpaths**
   - ◆ **Job Setup**
2. Enter values shown on the following dialog box.



3. Move the red arrow by moving the mouse to the left corner of the stock and click the left mouse button. The arrow will move automatically.
4. Choose **OK**.

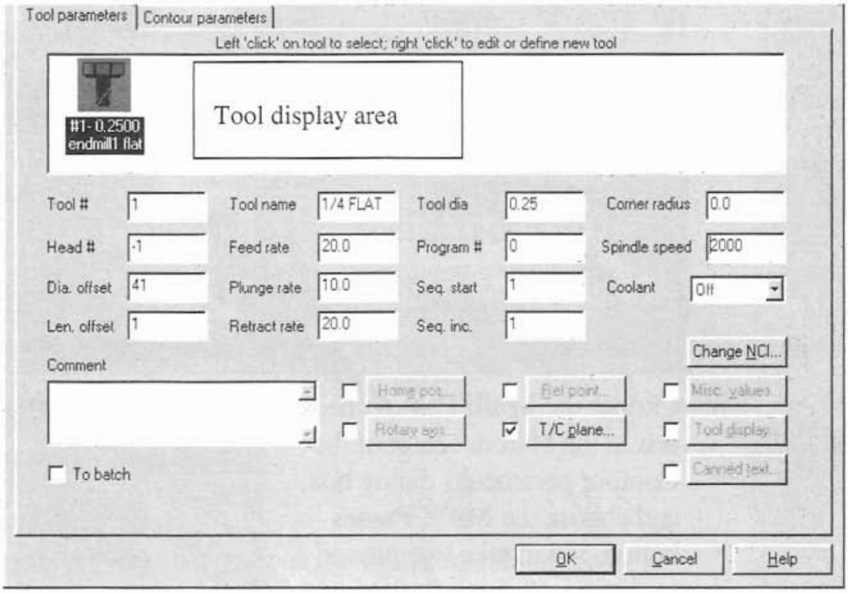
▶ *Chain the contour to be machined*

1. Choose
   - **MAIN MENU**
   - **Toolpaths**
   - **Contour**
2. Choose **Chain**.
3. Select a position on the line just above the bottom left radius as shown in the diagram to the right.
4. Choose **Done**.



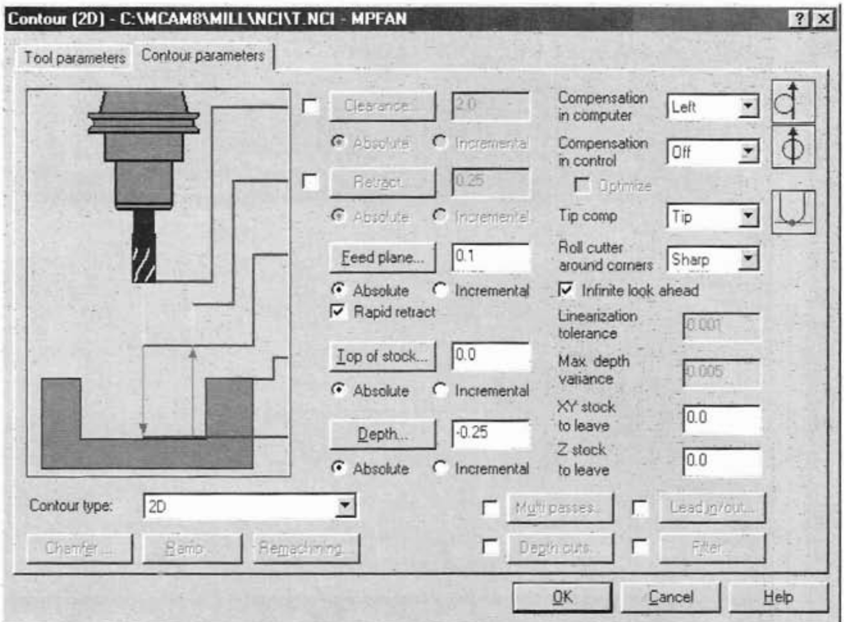
▶ *Set the parameters*

1. Right-click in the tool display area. A right-click menu displays.
2. Choose **Get tool from library**.
3. Choose the 1/4" flat end mill.
4. Choose **OK**.
5. Enter the values as shown on the following dialog box.



6. Select the **Contour parameters** tab at the top of the parameters dialog box.

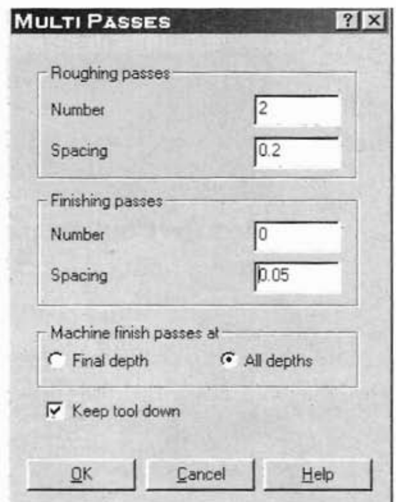
7. Enter the values as shown on the following dialog box.



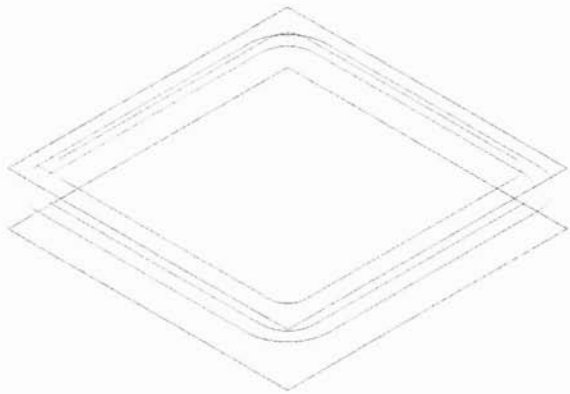
8. Choose the **Multi Passes** check box at the bottom center of the Contour parameters dialog box, and choose the **Multi Passes** button. It will take two passes to remove all the material from the corners.

9. Enter the values as shown in the dialog box at right.

10. Choose
   - **OK**
   - **OK**



11. A toolpath should appear in the graphics window and look like the following picture.



▶ *Checking the program by backplotting*

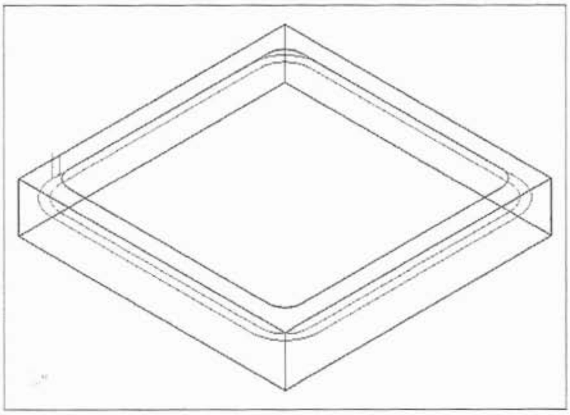
Backplotting displays the path the tool takes to cut the part and lets you catch errors in the program before you machine the part.

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Toolpaths**
2. Choose **Operations** to open the Operations Manager.
3. Choose **Backplot**.
4. Select options as shown at right. Click on **Show path** several times. As you do, you will see the letter to the right change from **Y** (for yes) to **N** (for no). Toggle (click) it so that the **Y** appears.

5. Choose **Step**.
6. Continue to press **Step** until the backplot is complete.

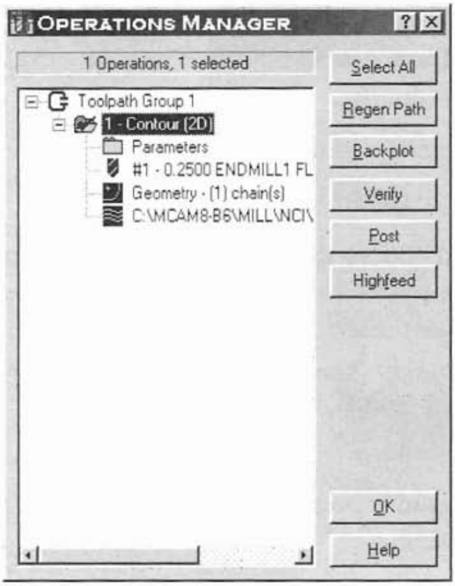
Note: The prompt area at the bottom of your screen displays the machining time for the part. Try checking the toolpath in a number of different views or angles, using icons from the toolbar.

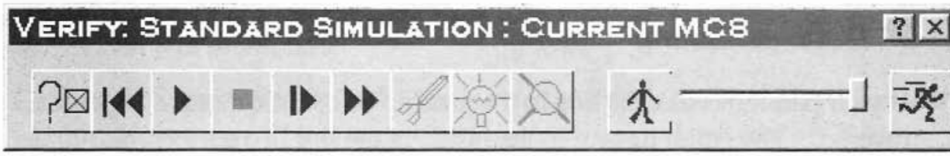
7. Press [Esc] to exit the Backplot menu. Your drawing should look like the following picture.



▶ *Verify the program*

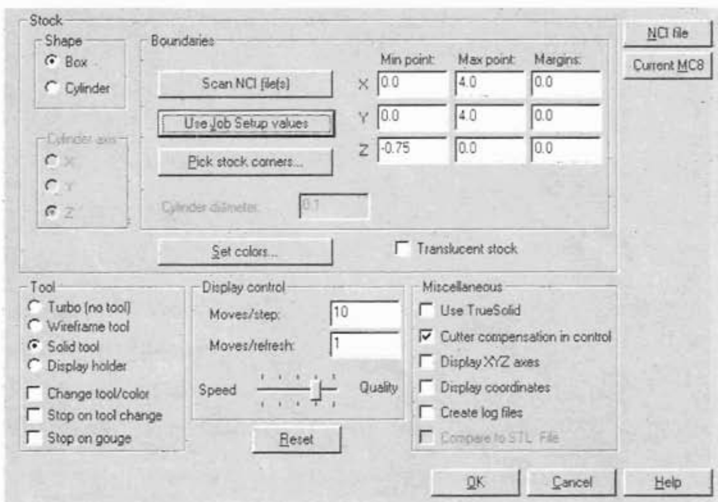
1. Choose the **Verify** button.





2. Choose the **Configure** button from the toolbar.

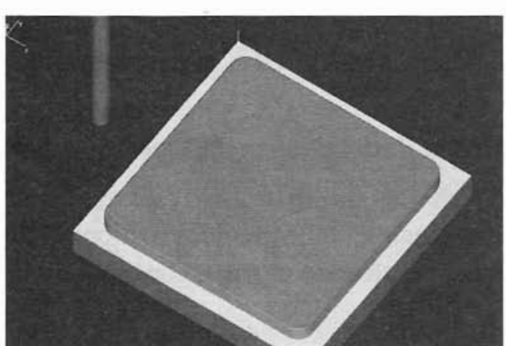
3. The Verify Configuration dialog box opens. Choose the **Use Job Setup values** button. Enter the values shown in the dialog box below.



4. Choose **OK**.

5. Choose the **Machine** button from the Verify toolbar.

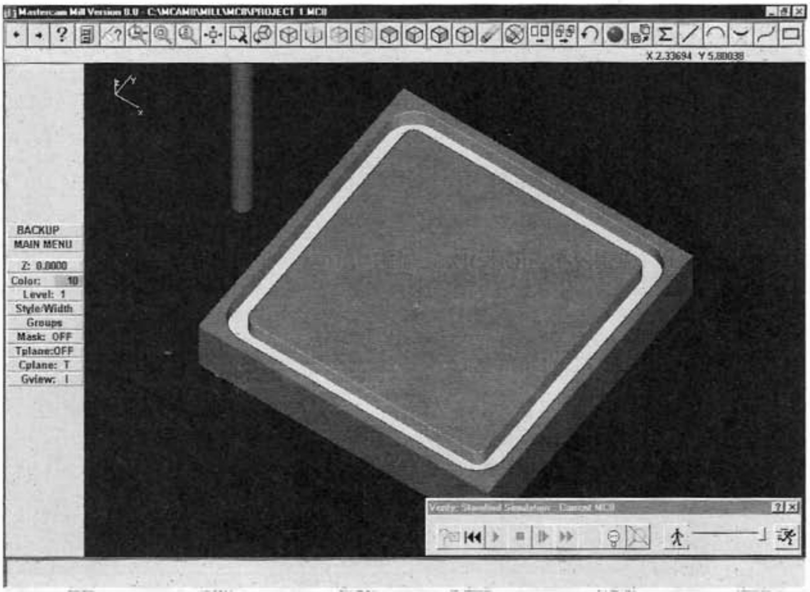
Note: It may take up to a minute to draw the screen. The verified part should look like the following picture.



▶ Troubleshooting

If your part looks like the one in the graphic below, the cutter compensation is reversed. The cutter has cut to the inside of the line. To correct this problem, change to cutter compensation in the computer using the following steps.

1. Close the Verification toolbar by selecting the **X** in the upper right corner of the Verification toolbar.

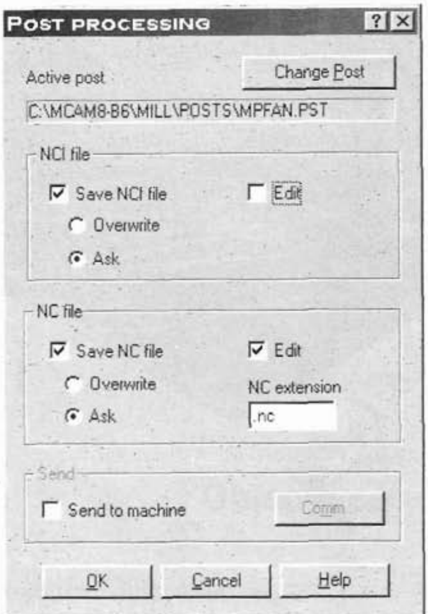


2. The Operations Manager opens. Select the **Parameters** icon under the contour toolpath.
3. Once in the Parameters screen, select the **Contour Parameters** tab.
4. Change **compensation in computer** option in the upper right. If it set to Left, change it Right. If it is set to Right or Off, change it to Left.
5. Choose **OK**. There is now a red X in the toolpath icon telling us it was changed.
6. Choose the **Regen Path** button in the Operations Manager. This updates the changes to your program.
7. Return to the beginning of the previous section on Verification and proceed with the verification process, checking for the changes.

▶ *Post the NC Code*

Note: Before continuing, ask your instructor if the correct post processor is set up in Mastercam. If it is not, skip to the last step and save the MC8 file again.

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Toolpaths**
   - ◆ **Post**
2. Enter the values shown in the following dialog box.

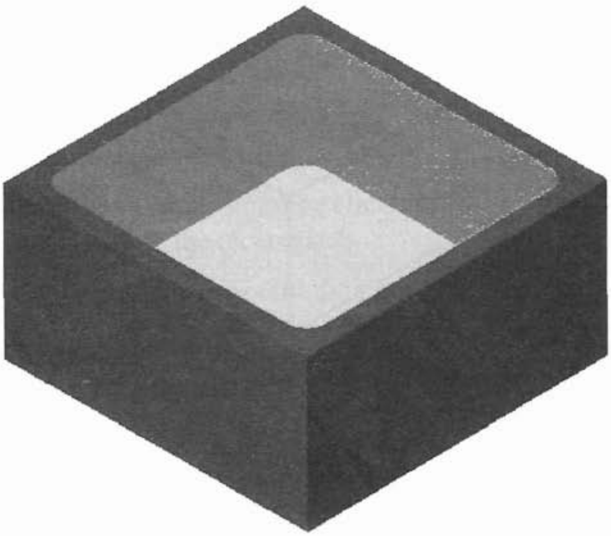


3. Choose
   - ◆ **OK**
   - ◆ **Save**
   - ◆ **Save**
4. The NC file will be displayed in a white editor window. Close the editor screen by selecting the **X** in the upper right corner.
5. Choose **OK** to close the Operations Manager.

▶ *Save the MC8 file again*

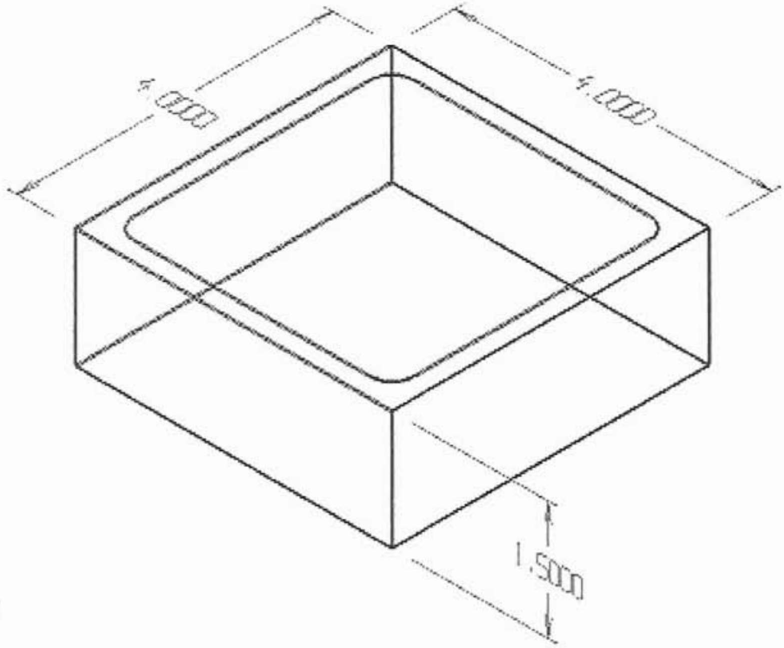
1. Choose
    - **MAIN MENU**
    - **File**
    - **Save**
    - **Save**
    - **Yes**

3 Jewel Box Base - Pocket



Objectives

- Navigate the Mastercam menus.
- Create lines and arcs.
- Translate and offset lines.
- Enter coordinates.
- Construct 3D geometry.
- Use chaining methods.
- Create a pocket toolpath.
- Enter parameters for a pocket toolpath.
- Perform solid model verification of the toolpath.



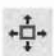
Geometry creation

▶ *Create the base of a jewel box*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Rectangle**
   - ◆ **2 points**
2. *Enter the coordinates.* **X0, Y0**

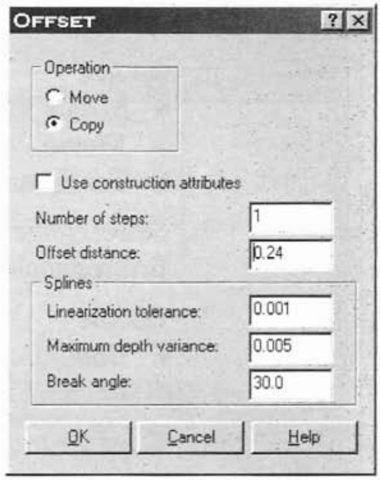
Note: Be sure to enter the number zero, not the letter "O".

3. *Enter the coordinates.* **X4, Y4**



4. Choose the **Fit** button on the toolbar to center the geometry.

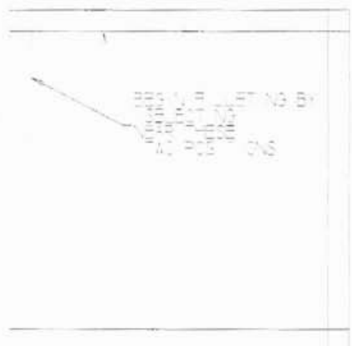
Create the inside entities



1. Choose
   - **MAIN MENU**
   - **Xform**
   - **Offset**
2. Enter the values shown in the dialog box at right.
3. Choose **OK**.
4. *Select the lines to offset.* Select any one of the lines of the rectangle.
5. Indicate the offset position by clicking anywhere inside the rectangle.
6. Repeat the same process for the three remaining lines.

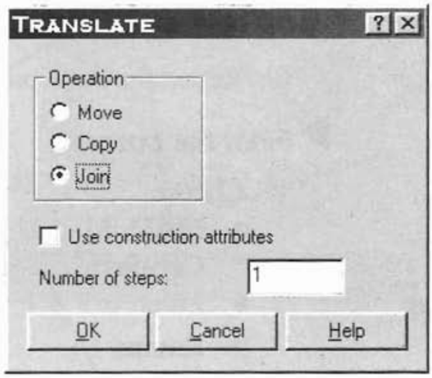
Fillet the corners

1. Choose
   - **MAIN MENU**
   - **Create**
   - **Fillet**
   - **Radius**
2. *Enter the fillet radius.* **0.26**
3. Select one of the two new lines that touch each other as shown in the diagram at right.
4. Select another new line that intersects with the first line.
5. Repeat the above steps for the remaining three corners.



▶ Make the drawing 3D

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Xform**
   - ◆ **Translate**
2. Select entities to translate by clicking on the four outside lines of the first rectangle.
3. Choose
   - ◆ **Done**
   - ◆ **Rectang**
4. *Enter the translation vector.* **Z-1.5**
5. Enter the values shown at right.
6. Choose **OK**.

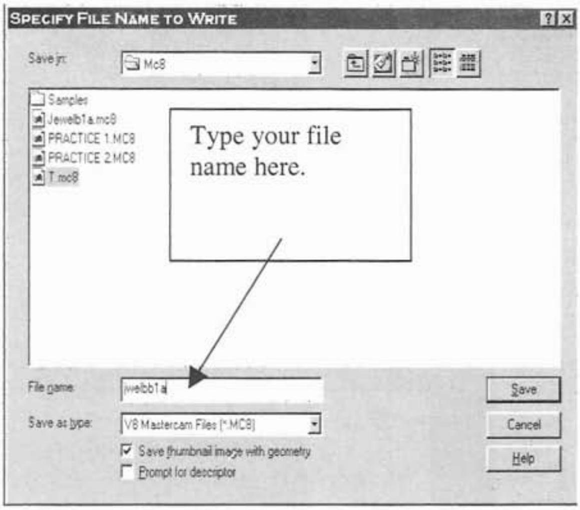


7. Choose the **Gview (isometric)** button from the toolbar to change the graphics view to Isometric.

8. Choose the **Fit** button from the toolbar to fit the geometry in the graphics window.

Save the drawing

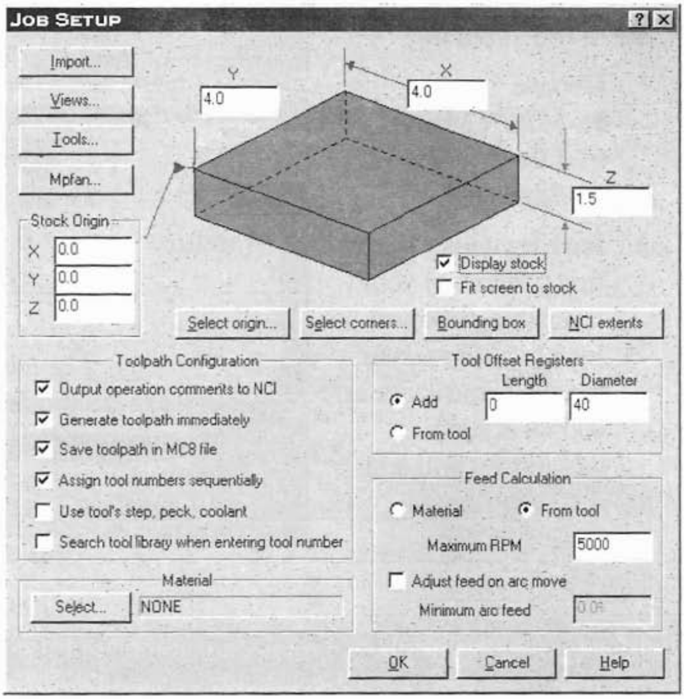
1. Choose
   - **MAIN MENU**
   - **File**
   - **Save**
2. Save by typing the first 6 letters of your last name, followed by 3 (eg:johnso3) in the file name space as shown at right. Mastercam will add the extension automatically.
3. Choose **Save**.



Toolpath creation

Define the rough stock

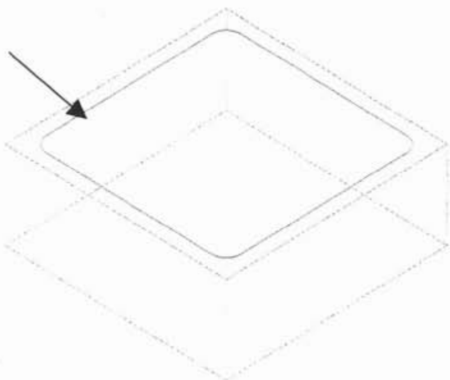
1. Press [Alt+J].
2. Enter values shown on the following dialog box.



3. Move the red arrow by moving the mouse to the left corner of the stock and clicking the left mouse button. The arrow will move automatically.
4. Choose **OK**.

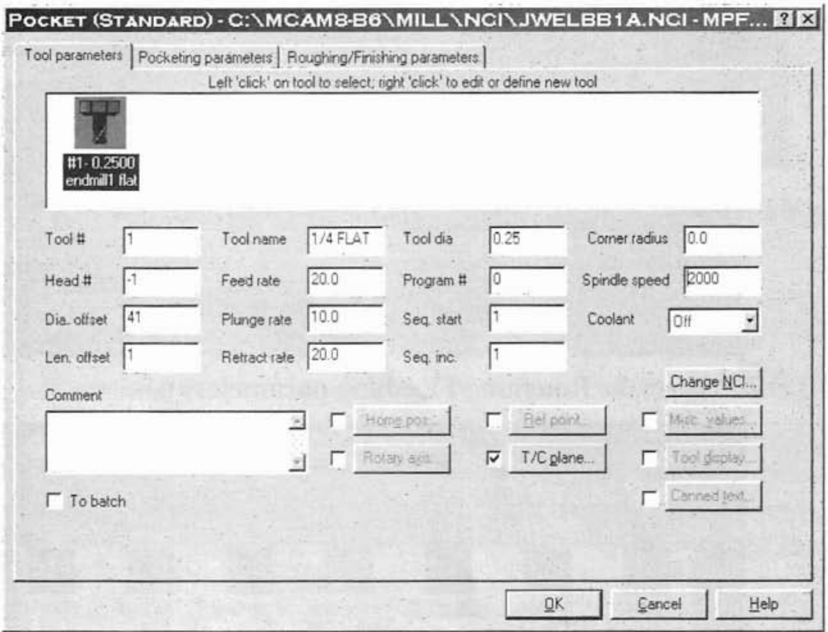
▶ *Chain the contour for machining*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Toolpaths**
   - ◆ **Pocket**
2. Select a position on the inside line just above the bottom left radius as shown in the diagram to the right.
3. Choose **Done**. A dialog box displays.

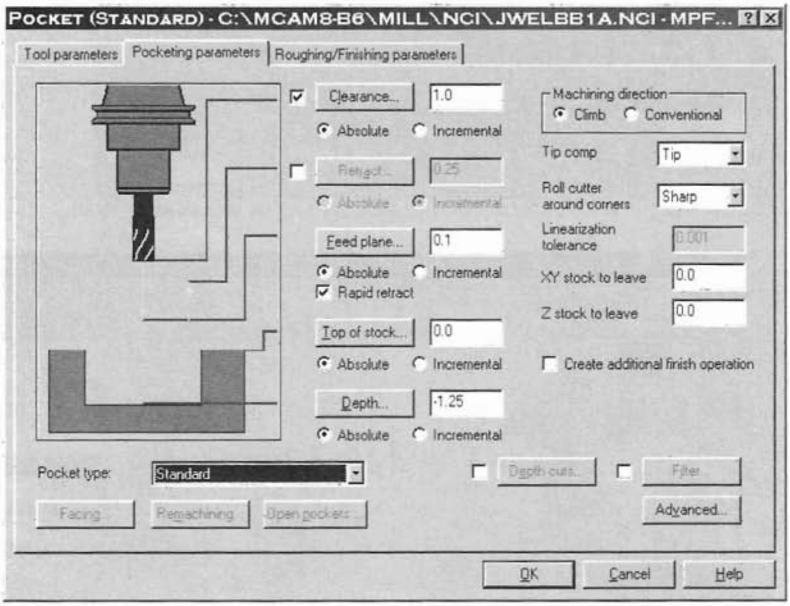


▶ Set the parameters

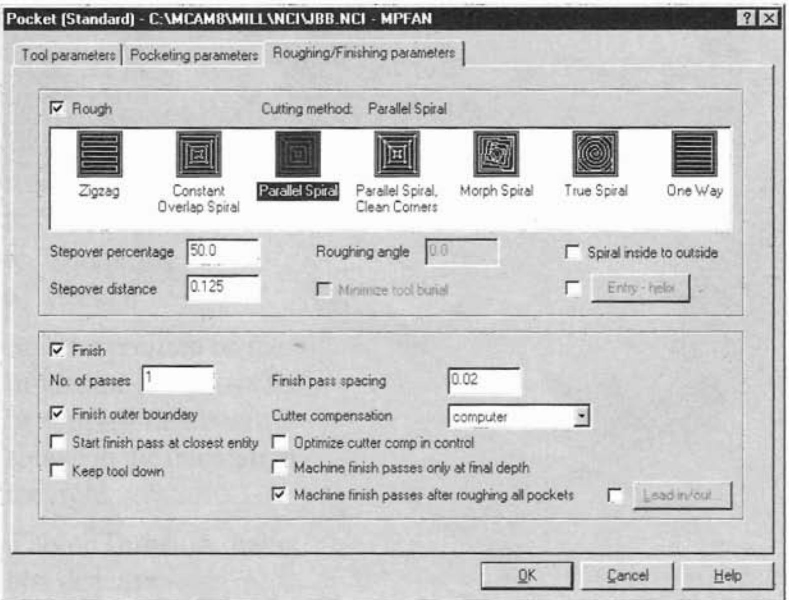
1. Right-click in the tool display area.
2. Choose **Get tool from library**.
3. Choose the ¼" flat end mill.
4. Choose **OK**.
5. Enter the values as shown in the following dialog box.



6. Select the **Pocketing parameters** tab. Enter the values shown in the following dialog box.



7. Select the **Roughing/Finishing parameters** tab.



8. Enter the values shown in the following dialog box.

9. Choose **OK**.

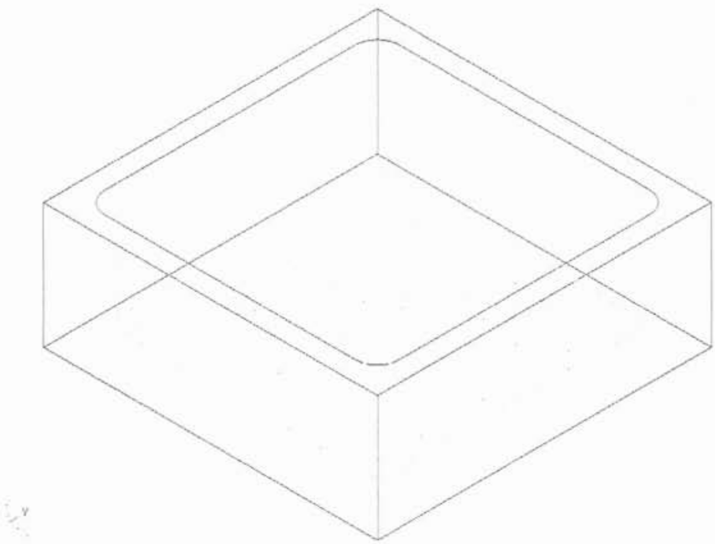
▶ *Check the program by backplotting*

Backplotting displays the path the tool takes to cut the part and lets you catch errors in the program before you machine the part.

1. Choose
   - **MAIN MENU**
   - **Toolpaths**
   - **Operations**
   - **Backplot**
2. Set the options as shown at right.
3. Choose **Run**.

Note: Try checking the toolpath in a number of different views or angles. Notice that the prompt area at the bottom of your screen displays the machining time for the part.

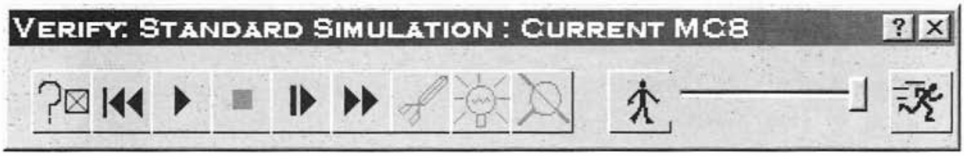
The toolpath should look like the following picture.



4. Press [Esc] to exit the Backplot menu. The Operations Manager dialog box displays.

▶ *Verify the program*

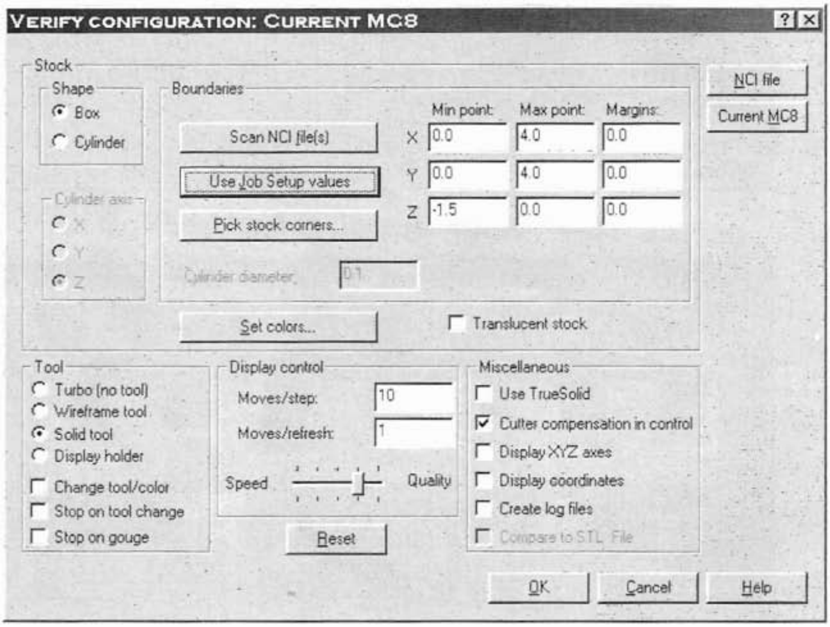
1. Choose the **Verify** button. The Verify toolbar opens.



2. Choose the **Configure** button on the toolbar.

3. The Verify Configuration dialog box opens. Choose the **Use Job Setup values** button.

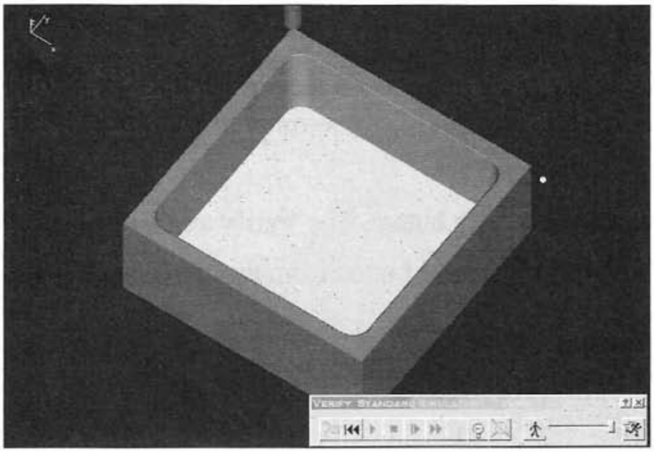
4. Enter the values shown in the following dialog box.



5. Choose **OK**.

6. Choose the **Machine** button from the Verify toolbar.

The verified part should look like the following picture.

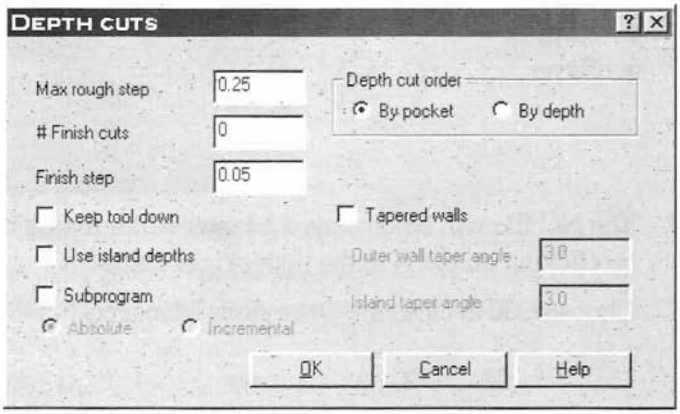


7. Close the Verify toolbar by clicking on the X in the upper-right corner.

▶ *Change the toolpath*

Note: The toolpath you created machines the pocket with one depth cut. This may not be suitable for the CNC machine or the tooling available in your classroom. This section shows you how to make a change in the toolpath, from one depth cut to three depth cuts.

1. Select the **Parameters** icon in the Operations Manager.
2. Select the **Pocketing parameters** tab.
3. Select the **Depth cuts** button.
4. Enter the values shown in the following dialog box.



5. Choose **OK**.

6. Choose **OK** in the Pocket Parameters dialog box. The Operations Manager displays.

Note: The toolpath icon has a red X through it. This tells us the toolpath has been modified and needs to be regenerated to activate the changes

7. Choose **Regen Path**.

8. Choose the **Verify** button. The Verify toolbar opens.

9. Choose the **Machine** button from the Verify toolbar.

Note: Watch for multiple depth passes during the verification process.

10. Close the Verify toolbar.

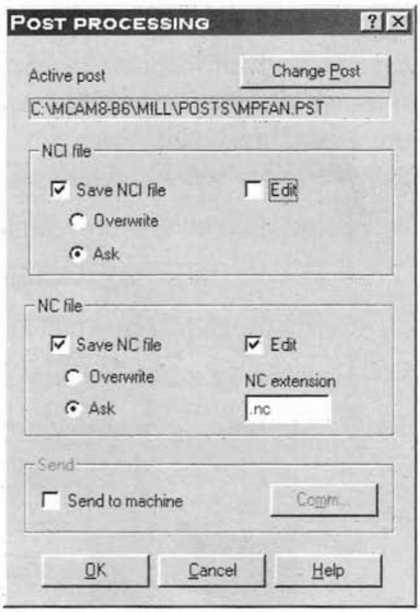
Post the code

Note: Before continuing, ask you instructor if the correct post processor is set up in Mastercam. If it is not, skip to the next step and save the file.

1. Choose **Post**.

2. Enter the values shown in the dialog box at right.

3. Choose
   - **OK**
   - **Save**
   - **Save**



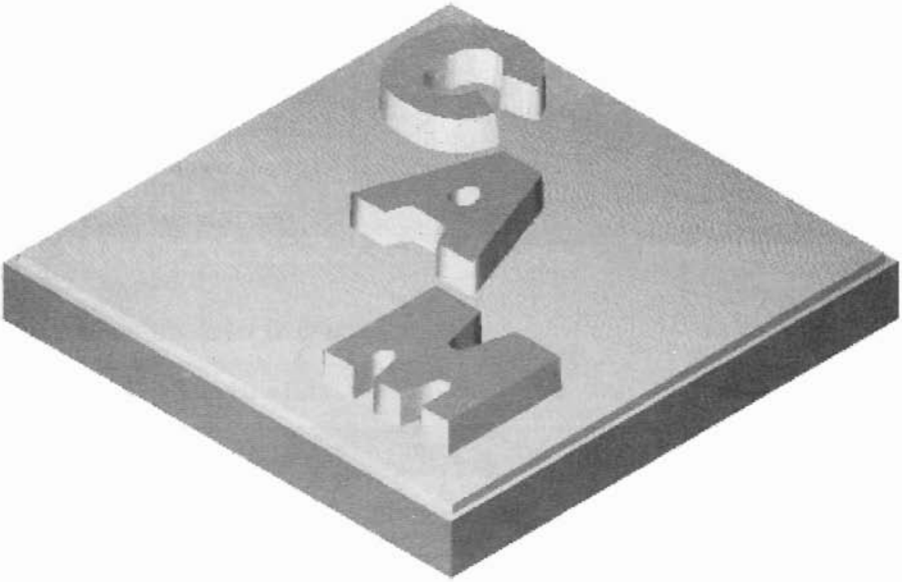
4. The NC file will be displayed a larger editor dialog box. Close the box by clicking on the X in the upper-right corner.

5. Choose **OK** to close the Operations Manager.

▶ *Save the MC8 file again.*

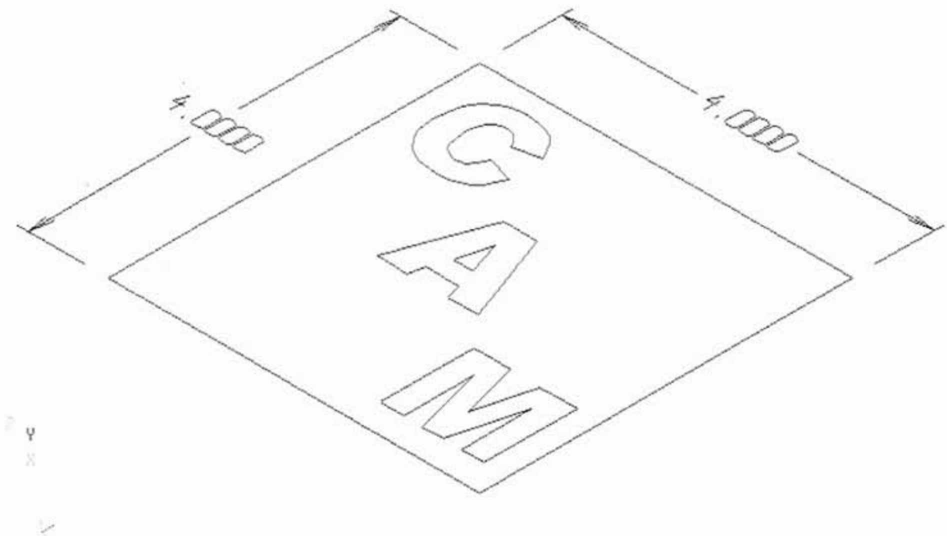
1. Choose
   - **MAIN MENU**
   - **File**
   - **Save**
   - **Save**
   - **Yes**

4 Jewel Box Lid - Top



Objectives

- Navigate the Mastercam menus.
- Create lines and arcs.
- Translate and offset lines.
- Enter coordinates.
- Construct 3-dimensional geometry.
- Use chaining methods.
- Create a pocket toolpath.
- Enter parameters for pocket and contour toolpaths.
- Perform solid model verification of the toolpath.
- Post the NC file.

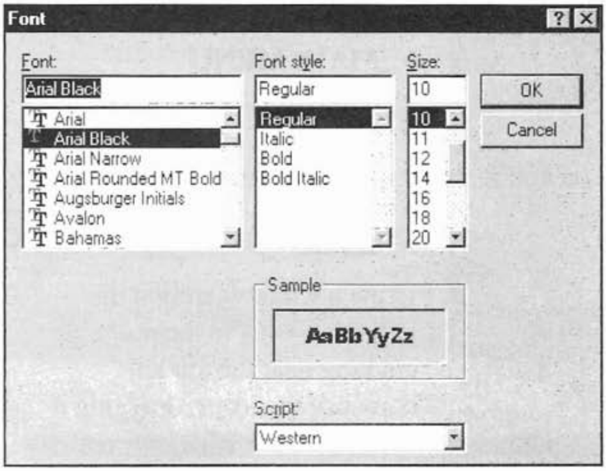


Geometry creation

▶ *Create a rectangle*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Rectangle**
   - ◆ **2 points**
2. *Enter the coordinates.* **X0, Y0**
3. *Enter the coordinates.* **X4, Y4**
4. Choose the **Fit** button on the toolbar to center the geometry.

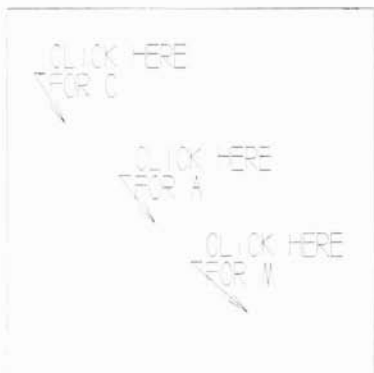
▶ *Create the letters*



1. Choose
    - ◆ **MAIN MENU**
    - ◆ **Create**
    - ◆ **Next Menu**
    - ◆ **Letters**
    - ◆ **True Type®**
2. Choose Arial Black as shown in the dialog box at right.
3. Choose **OK.**
4. *Enter the letter.* **C**

Note: Use capital letters.

5. *Enter the letter height.* **1**
6. Choose **Horizontal.**

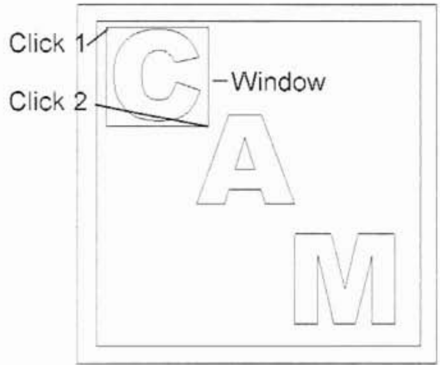


7. Position the cursor near the top left corner of the screen. Once the cursor is positioned, click the left mouse button. The letter will appear to the top and right of the cursor.
8. Choose
    - ◆ **True Type**
    - ◆ **Arial Black**
    - ◆ **OK**
9. *Enter the letter.* **A**
10. *Enter the letter height.* **1**
11. Choose **Horizontal.**
12. Position the cursor near the middle of the screen, clicking the mouse.
13. Repeat steps 6 through 10, positioning **M** near the bottom right.

Note: If the letters do not appear in the correct spots, you have two options.

▶ Option 1: Move the letter or letters

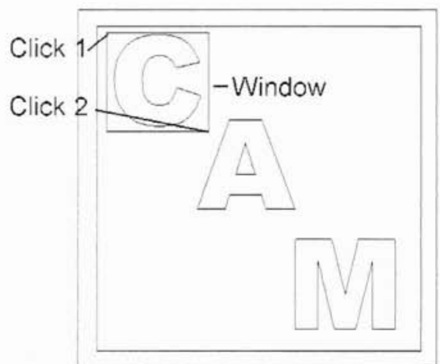
1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Modify**
   - ◆ **Drag**
   - ◆ **Translate**
   - ◆ **Window**
   - ◆ **Inside**



2. Draw a window around the letter you wish to move by clicking near the top left corner of the letter, dragging it to the bottom right corner of the letter and clicking again.
3. Choose **Done**.
4. Enter a starting point by putting your cursor near the letter you wish to move until the **Sketch** option is highlighted on the menu.
5. Click on the letter and drag it to the correct position. Click when finished.
6. To drag other letters, repeat the steps above.

▶ Option 2: Delete and recreate letters

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Delete**
   - ◆ **Window**



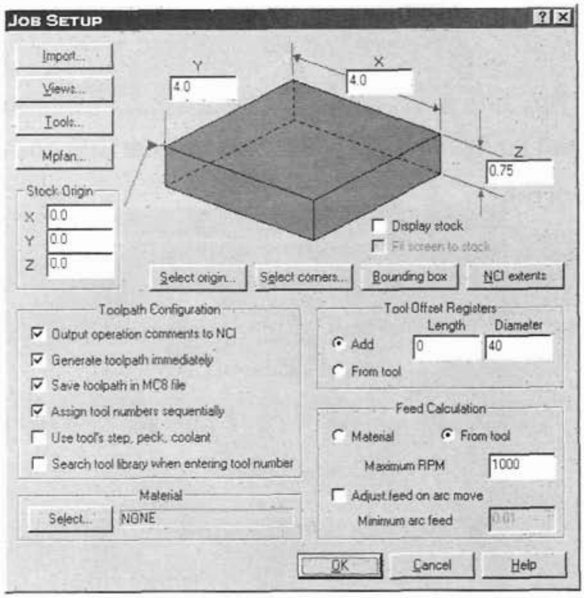
2. Draw a window around the letter(s) you wish to remove. Click in the top left corner, drag the mouse to the bottom-right corner and click again.
3. Repeat steps in the **Create the letters** on page 55 to recreate a letter.

▶ *Save the drawing*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **File**
   - ◆ **Save**
2. In the File Name box, type the first 6 letters of your last name, followed by 4 (for example, johnso4).
3. Choose **Save**.

Toolpath creation

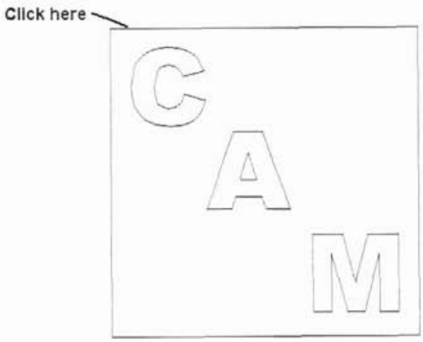
▶ *Define the rough stock*



1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Toolpaths**
   - ◆ **Job Setup**
2. Enter values shown in the dialog box at right
3. Move the stock origin arrow by selecting the left corner of the stock, and clicking your mouse.
4. Choose **OK**.

▶ *Chain the pocket*

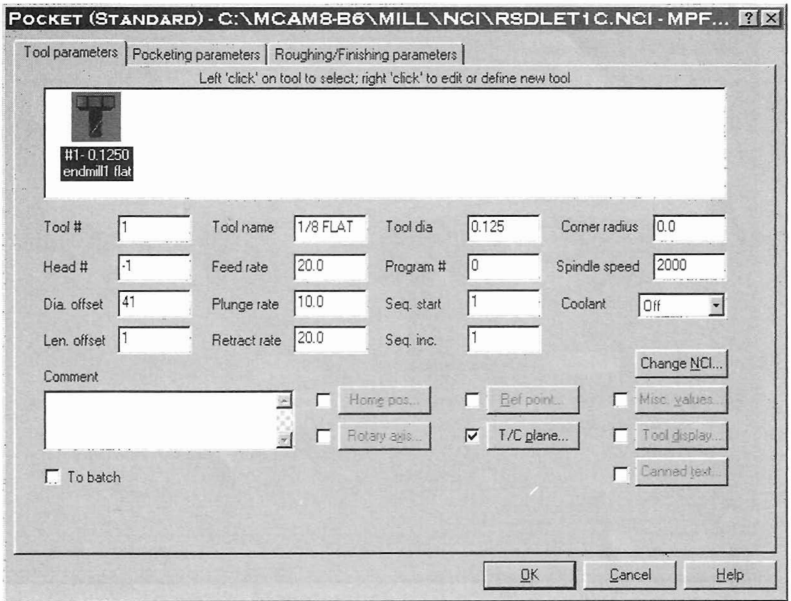
1. Choose **Pocket**.
2. Click on the upper left corner of the rectangle as shown in the following diagram.



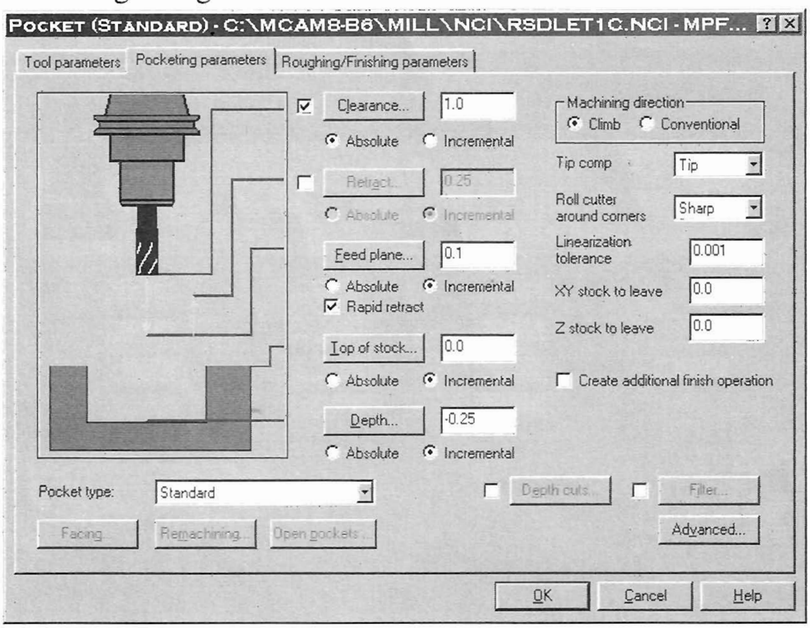
3. Choose
   - **Mode**
   - **Window**
4. Draw a window around the letters. Keep the window inside the rectangle.
5. Select a point inside one of the letters.
6. Choose **Done**.

▶ *Set the parameters*

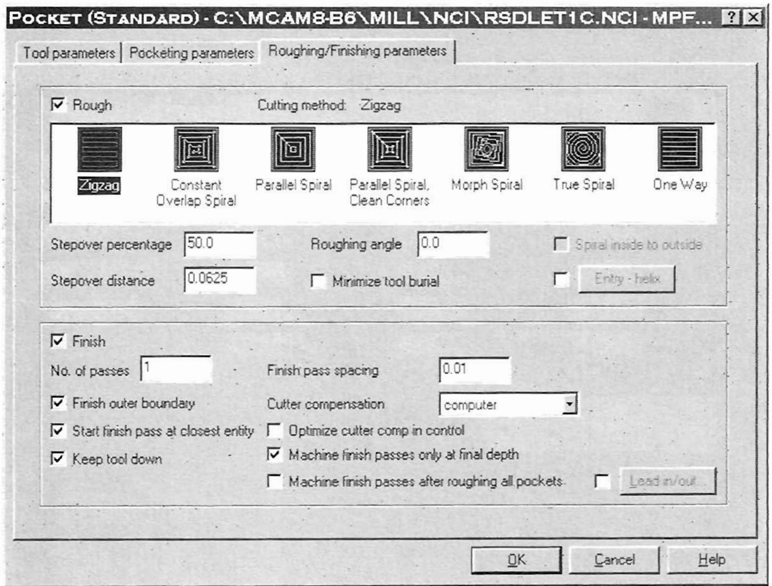
1. Right-click in the tool display area.
2. Choose **Get tool from library**.
3. Choose the 1/8" flat endmill.
4. Choose **OK**.
5. Enter the values shown on the following dialog box.



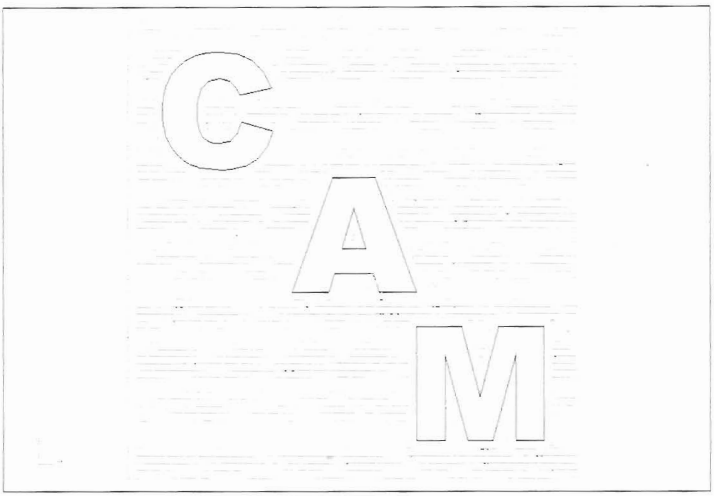
6. Select the **Pocketing parameters** tab and enter the values shown on the following dialog box.



7. Select the **Roughing/Finishing parameters** tab and enter the values shown on the following dialog box.



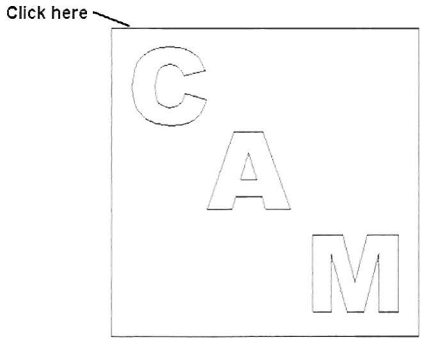
8. Choose **OK**. The toolpath should look like the following picture:



Chain the contour

1. Choose
   - **MAIN MENU**
   - **Toolpaths**
   - **Contour**

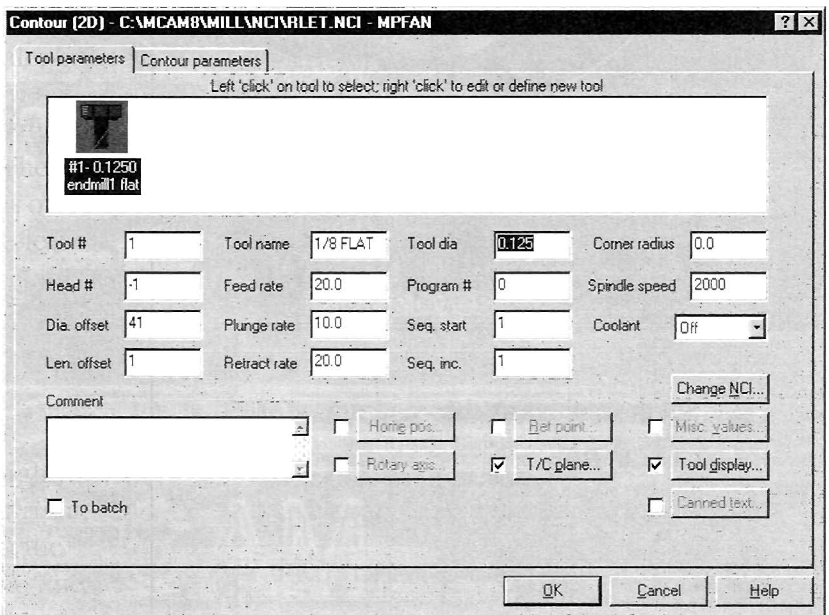
2. Select the top left corner of the contour as shown in the following drawing.



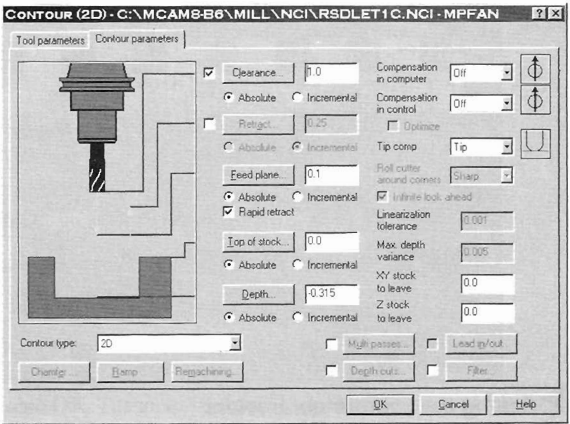
3. Choose **Done**.

▶ *Setting the contour parameters*

1. The 1/8" flat endmill is already selected. Enter the remaining values shown in the following dialog box.



2. Select the **Contour parameters** tab and enter the following values.



Note: It's important that the Compensation in computer is off and Compensation in control is off so the center of the tool will go right down the top of the letters.

3. Choose **OK**.

▶ *Check the program by backplotting*

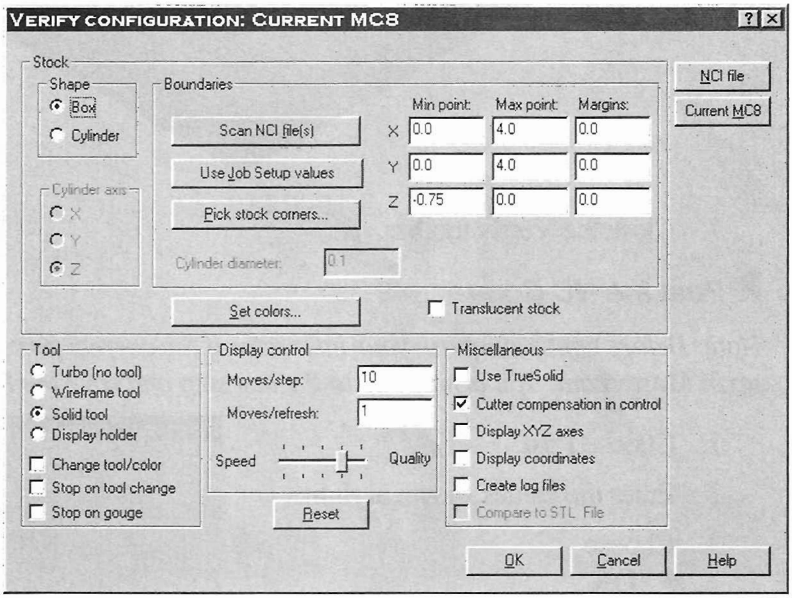
1. Choose
   - ◆ **Operations**
   - ◆ **Select All**
   - ◆ **Backplot**

2. Set the options as shown at right.
3. Choose **Run**.

▶ *Verify the program*

1. Choose
   - **BACKUP**
   - **Select All**
   - **Verify**

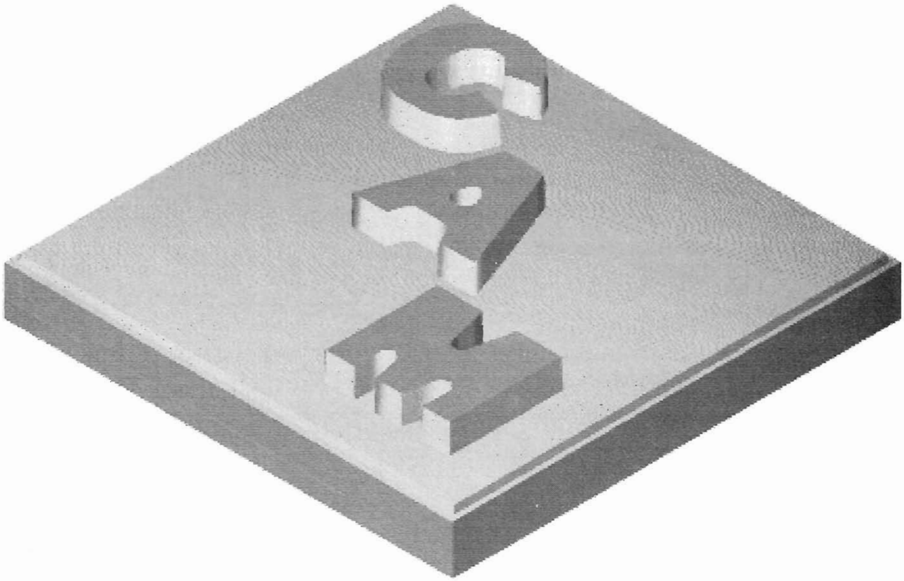
2. Choose the **Configure** button from the toolbar.
3. The Verify Configuration dialog box displays. Select the **Use Job Setup values** button.
4. Enter the values shown in the following dialog box.



5. Choose **OK.**

6. Choose the **Machine** button from the Verify toolbar.

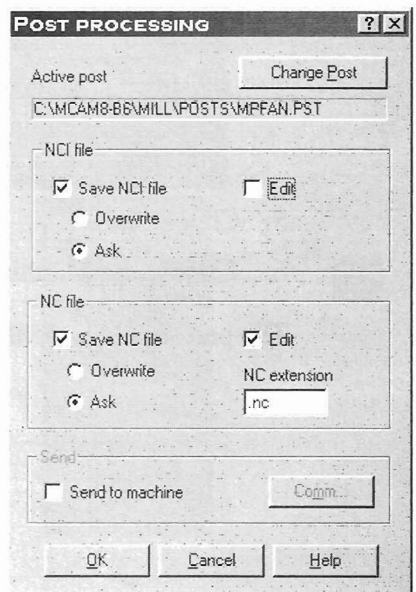
The part should look like the following picture.



7. Close the Verify toolbar.

Post the NC Code

Note: Before continuing, ask your instructor if the correct post processor is set up in Mastercam. If it is not, skip to the last step and save the MC8 file.



1. Choose **Post**.
2. Enter the values shown at right.
3. Choose
   - **OK**
   - **Save**
   - **Save**
4. The NC file is displayed in an editor window. Close the editor by selecting the **X** in the upper right corner.
5. Choose **OK** to exit the Operations Manager.

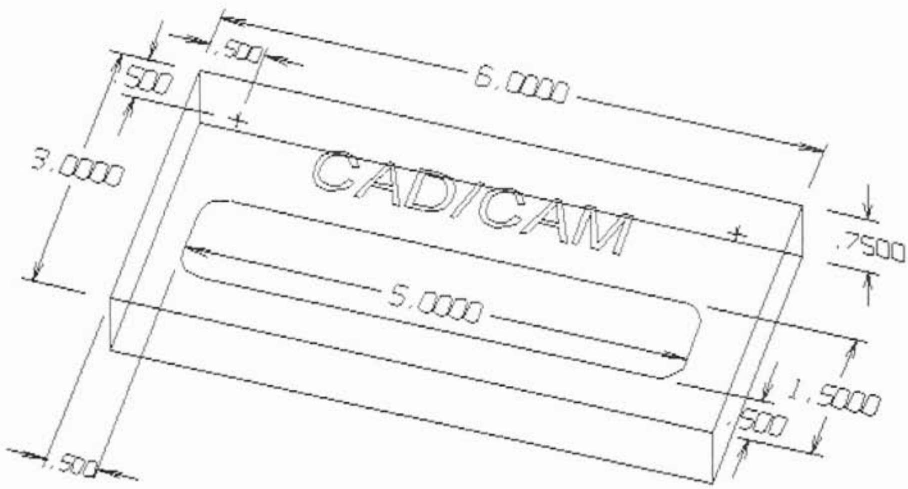
▶ *Save the MC8 file again*

1. Choose
   - **MAIN MENU**
   - **File**
   - **Save**
   - **Save**
   - **Yes**

5 Pen Holder

Objectives

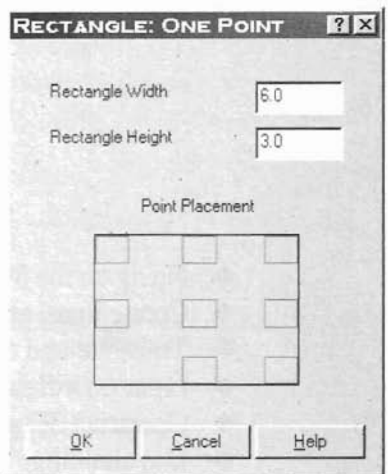
- Navigate the Mastercam menus.
- Create lines, arcs and points.
- Translate and offset lines.
- Enter coordinates.
- Construct 3D geometry.
- Use chaining.
- Create contour, pocket, and drill toolpaths.
- Enter parameters for contour, pocket, and drill toolpaths.
- Perform solid model verification of the toolpath.
- Post the NC file.



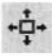
Geometry creation

▶ *Create a rectangle*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Rectangle**
   - ◆ **1 point**
2. Enter the values shown in the dialog box at right, including width, height and point placement.
3. Choose
   - ◆ **OK**
   - ◆ **Origin**
4. Press [Esc] to exit the function.

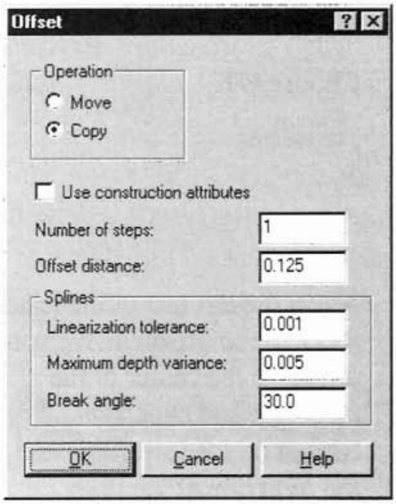


▶ *Create the inside entities*



1. Choose the **Fit** button on the toolbar to center the geometry.

2. Choose
   - **MAIN MENU**
   - **Xform**
   - **Offset**



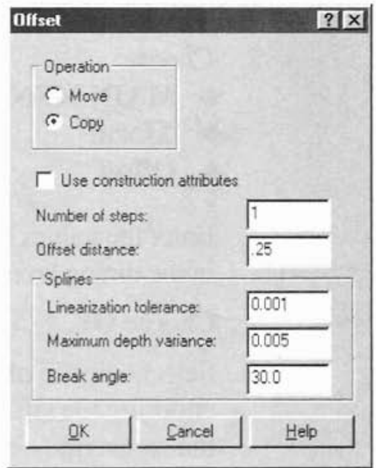
3. Enter the values shown in the dialog box.
4. Choose **OK**.
5. Select a line to offset by choosing the left line in the rectangle.
6. Indicate the offset position by clicking anywhere inside the part.
7. Repeat the same process for the three remaining lines.
8. Press [Esc] to exit the function.

▶ *Trim the corners of the new lines*

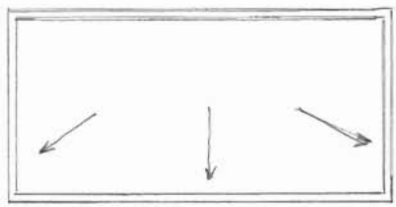
1. Choose
   - **MAIN MENU**
   - **Modify**
   - **Trim**
   - **2 entities**

2. With the cursor, select the two inside lines as shown in the diagram at right.
3. Select the remaining 3 corners to trim.
4. Press [Esc].

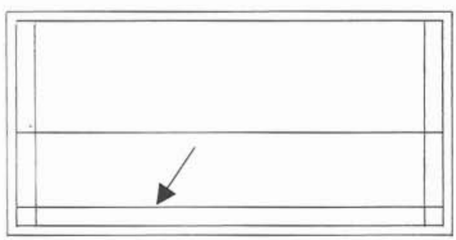
▶ Create a rectangle for the pocket



1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Xform**
   - ◆ **Offset**
2. Enter the values shown in the dialog box.
3. Choose **OK**.
4. Select the left line of the inside rectangle to offset. Move your mouse to the center of the rectangle and click again. Repeat on the lower line and the right line of the inner rectangle.

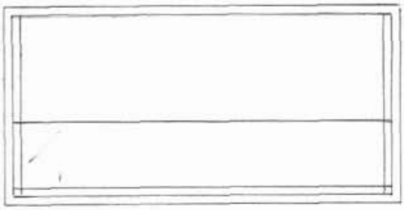


5. Choose
   - ◆ **BACKUP**
   - ◆ **Offset**
6. Enter **1.0** as the offset distance in the dialog box.
7. Choose **OK**.
8. Select the bottom inside line to offset, clicking on a point near the center of the line.

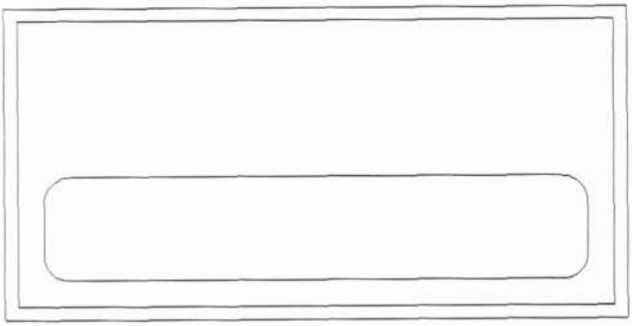


▶ Fillet the corners of the new pocket

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Fillet**
   - ◆ **Radius**
2. *Enter the radius.* **0.275**
3. Create the first fillet by selecting two intersecting lines as shown in the diagram at right.



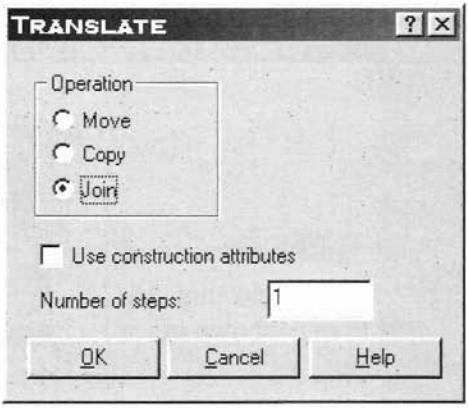
4. Create the remaining 3 fillets by selecting the intersecting lines in the other three corners of the rectangle just created. Your drawing should look like the following picture.



▶ Make the drawing 3D

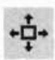
1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Xform**
   - ◆ **Translate**
2. Select the four lines that make up the outermost rectangle.
3. Choose
   - ◆ **Done**
   - ◆ **Rectang**
4. *Enter the translation vector.* **Z-0.75**

5. Enter the values shown on the following dialog box.

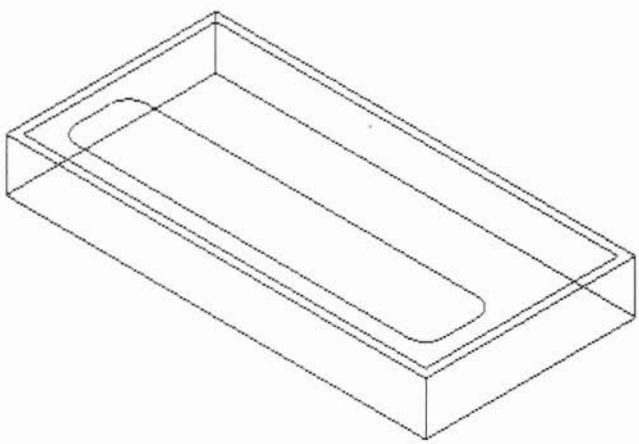


6. Choose **OK**.

7. Choose the **Gview (isometric)** button from the toolbar.



8. Choose the **Fit** button from the toolbar to fit the geometry.



Create points for drill holes

1. Choose the **Gview (Top)** button from the toolbar to change the graphics view to Top.

2. Choose the **Fit** button from the toolbar to fit the geometry in the graphics window.
3. Choose
   - **MAIN MENU**
   - **Create**
   - **Point**
   - **Position**
4. *Enter coordinates.* **X0.375, Y2.625**
5. *Enter coordinates.* **X5.625, Y2.625**

Create letters for engraving

1. Choose
   - **MAIN MENU**
   - **Create**
   - **Next menu**
   - **Letters**
   - **True Type®**
2. Select a font to be used for engraving. (Arial is used in the following pictures.)
3. Choose **OK**.
4. *Enter letters.* **CAD/CAM**
5. *Enter letter height.* **0.50**
6. Choose **Horizontal**.
7. *Enter letter spacing.* **0.125**
8. *Enter coordinates.* **X1.125, Y2.00**
9. The part should look similar to the picture at right.

Note: Use the Modify, Drag function described on page 56 if you need to adjust the letters.

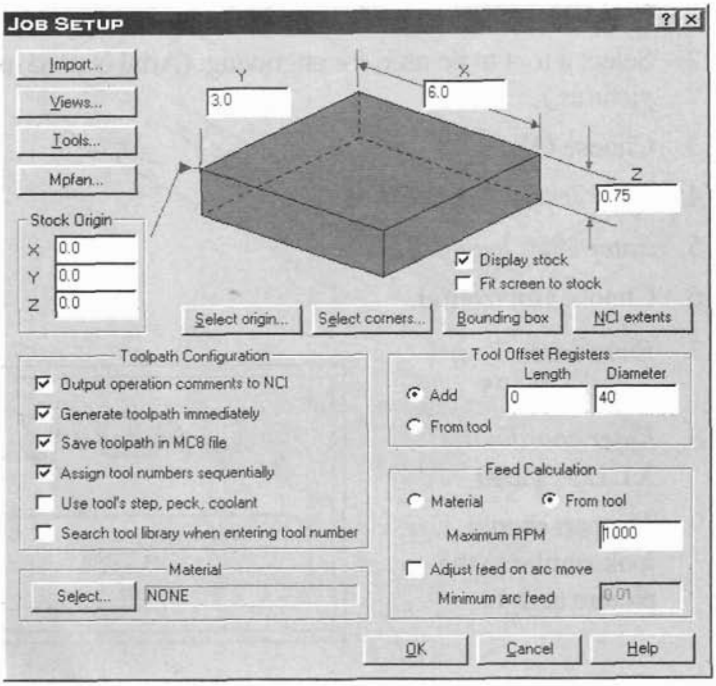
Save the drawing

1. Choose
   - **MAIN MENU**
   - **File**
   - **Save**
2. In the File name box, type the first 6 letters of your last name, followed by the number 5 (example: johnso5).
3. Choose **Save**.

Toolpath creation

Define the rough stock

1. Choose
   - **MAIN MENU**
   - **Toolpaths**
   - **Job Setup**
2. Enter the values shown in the following dialog box.



3. Move the arrow from the center of the box in the Job Setup dialog box by clicking and releasing your mouse in the left corner of the stock.
4. Choose **OK**.

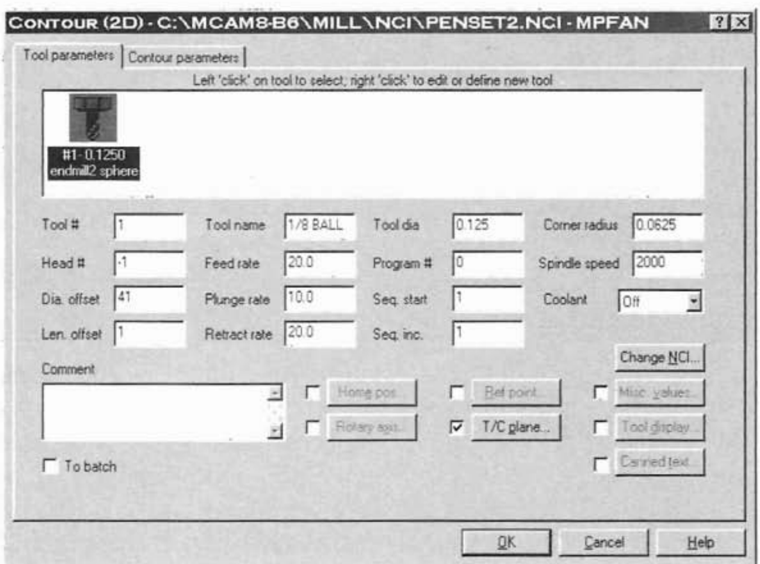
Note: This part can be cut with 1 or 2 tools. Ask your instructor if the CNC machine in your class is capable of an easy tool change. If it is not, then use 1 tool to do all the toolpaths.

▶ *Chain the contour to be machined*

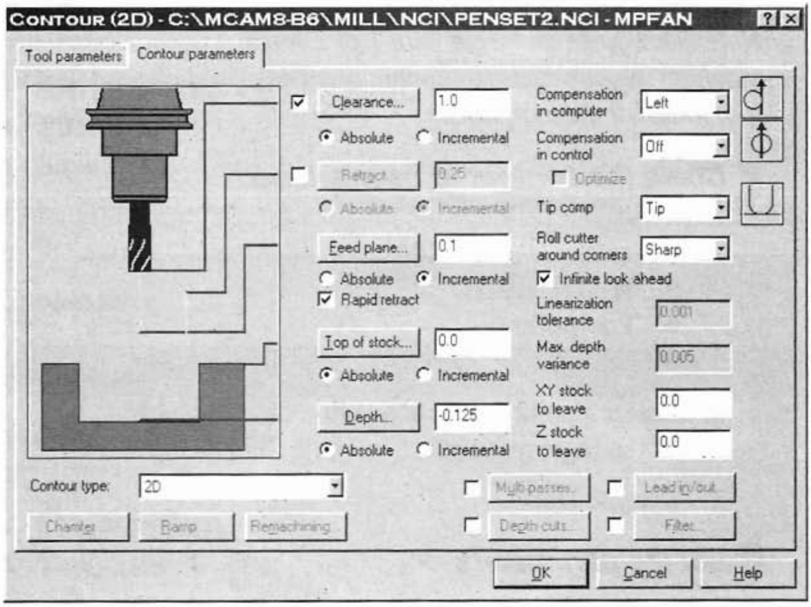
1. Choose
   - **MAIN MENU**
   - **Toolpaths**
   - **Contour**
2. Select a point on the inside line of the large rectangle.
3. Choose **Done**.

▶ *Set the parameters*

1. Right-click in the tool display area.
2. Choose **Get tool from library**.
3. Select the 1/8"spherical endmill (ball).
4. Enter the values shown in the following dialog box.



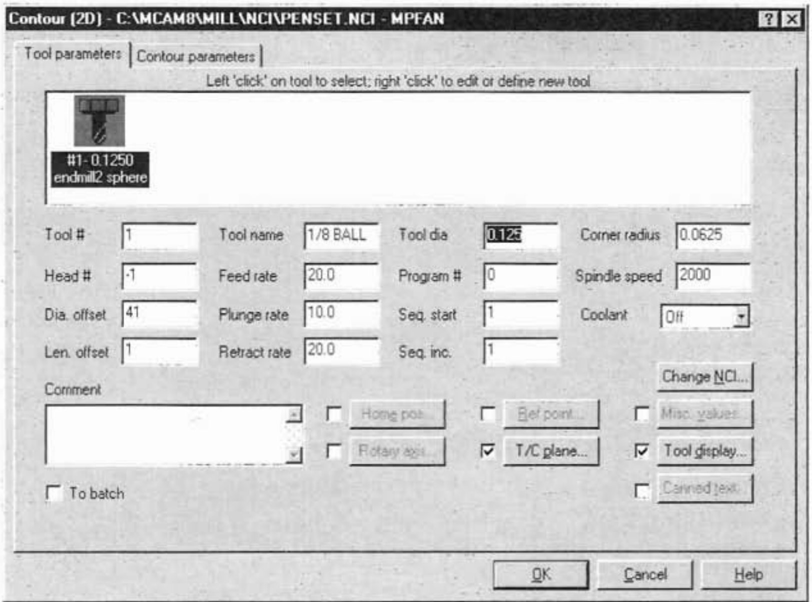
5. Choose the **Contour parameters** tab.
6. Enter the values shown in the following dialog box.



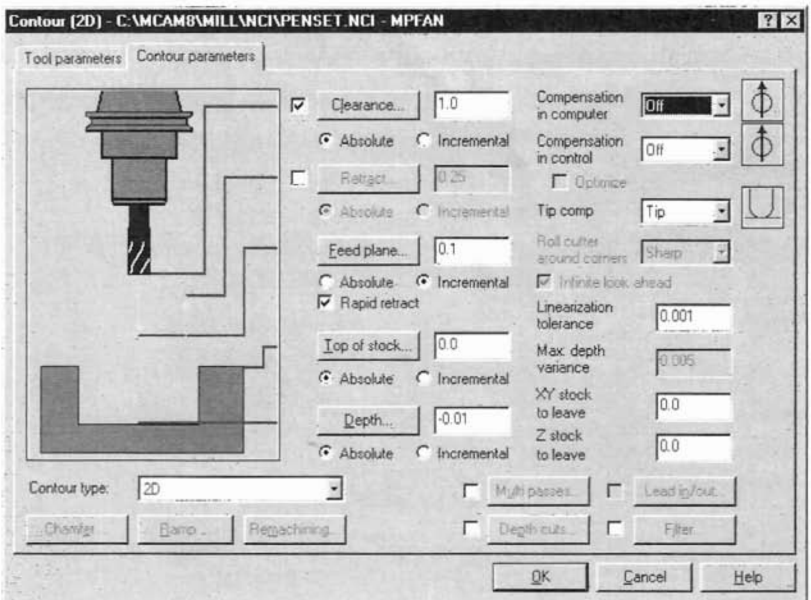
7. Choose **OK**.

Chaining the letters

1. Choose
   - **Contour**
   - **Window**
2. Draw a window around the letters.
3. Select a point where you wish cutting to begin inside one of the letters.
4. Choose **Done**.
5. Enter the values shown in the following dialog box.



6. Select the **Contour parameters** tab and enter the values shown.



7. Choose **OK**.

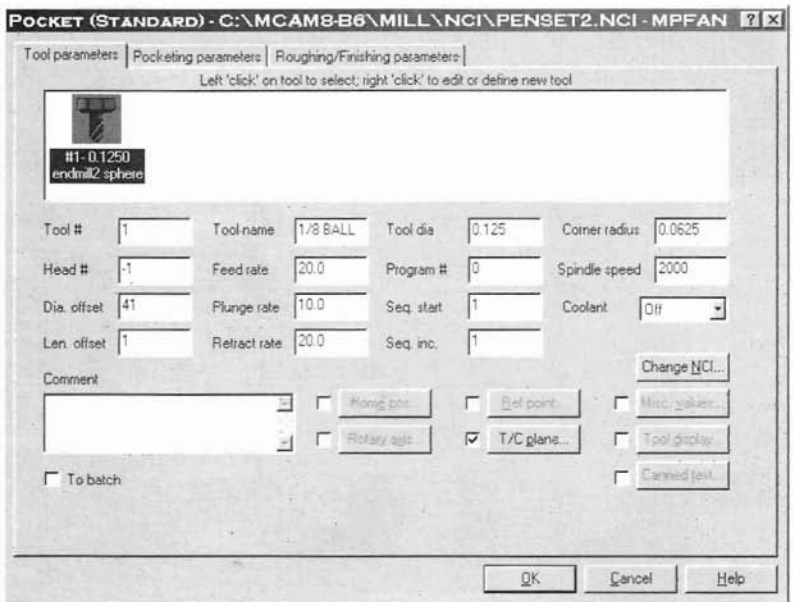
▶ Chain the pocket

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Toolpaths**
   - ◆ **Pocket**
   - ◆ **Chain**

2. Select a point on the line inside the pocket as shown at right.
3. Choose **Done**.

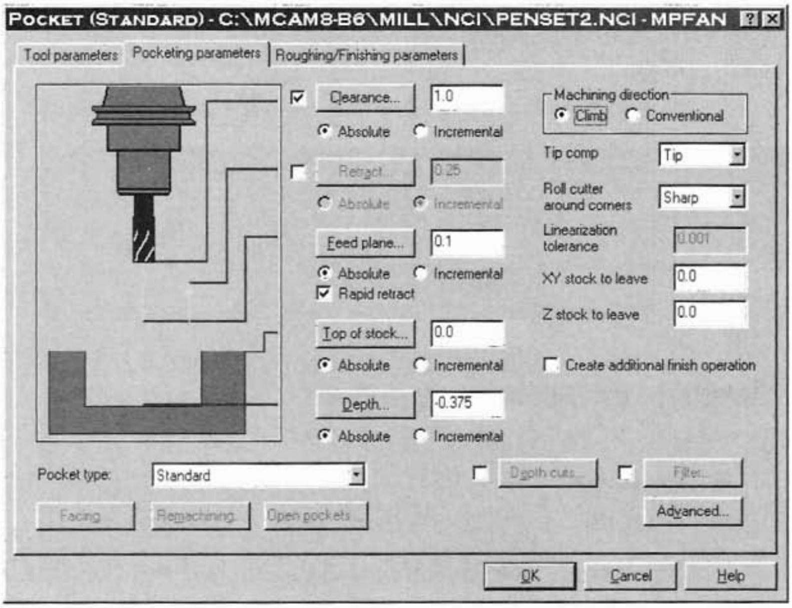
▶ Setting the pocket parameters

1. Enter the values shown on the following dialog box.



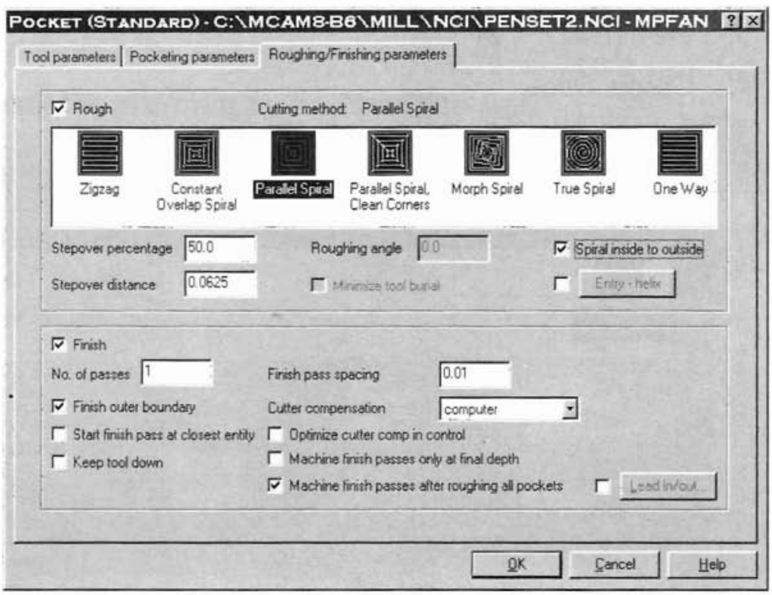
2. Select the **Pocketing parameters** tab.

3. Enter the values shown on the following dialog box.



4. Select the **Roughing/Finishing** tab.

5. Enter the values shown on the following dialog box.



6. Choose **OK**.

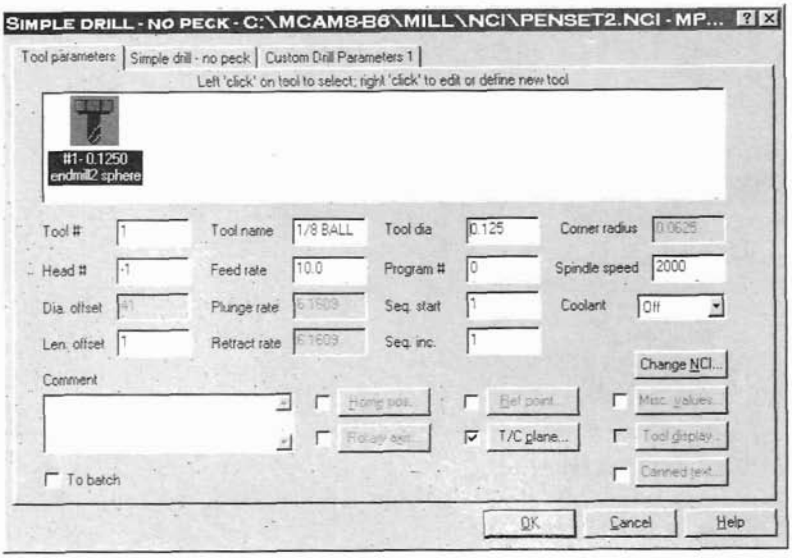
The toolpath should look like the following picture.

▶ *Selecting the holes to be drilled*

1. Choose
   - ◆ **Drill**
   - ◆ **Manual**
2. Select the points to drill found to the left and right of the CAD/CAM letters. As soon as the cursor is on top of one, Mastercam's AutoCursor will snap onto it.

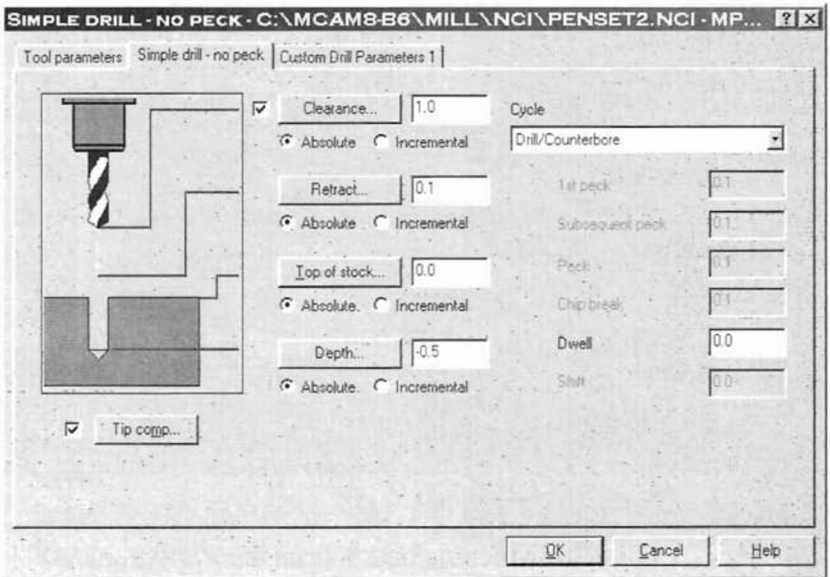
Note: Point will be automatically highlighted on the MAIN MENU.

3. Press [Esc].
4. Choose **Done**.
5. Enter the values shown in the following dialog box.



Note: A second tool can be added in this step. Ask the instructor if the CNC machine can do an easy tool change. If it can, select a .125 drill from the tool library. If it can't, you may have to use the .125 sphere mill to drill with.

6. Select the **Simple drill** tab.
7. Enter the values shown in the following dialog box.



8. Choose **OK**.

▶ *Check the program by backplotting*

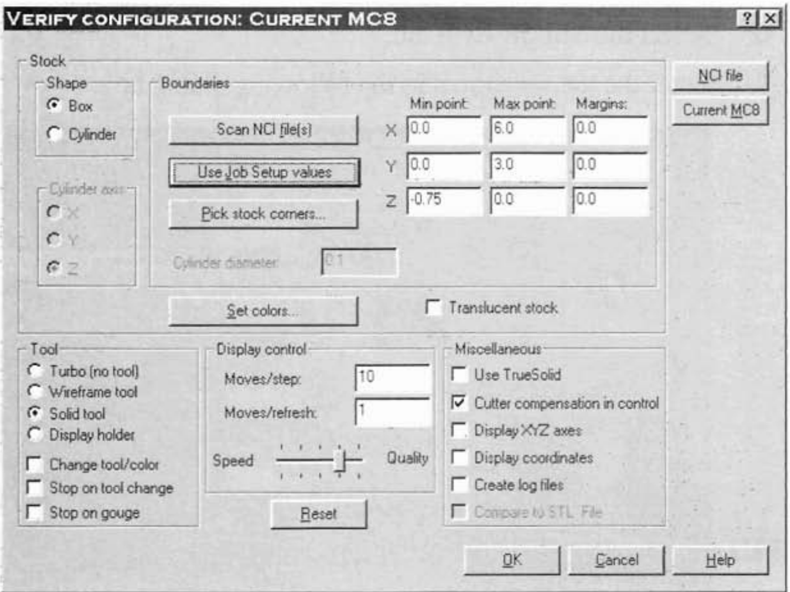
1. Press [Alt+O] to open the Operations Manager.
2. Choose
   - **Select All**
   - **Backplot**
   - **Run**
3. Press [Esc] to exit the backplot menu.

▶ *Verify the program*

1. Choose the **Verify** button from the Operations Manager.

2. Choose the **Configure** button from the toolbar.

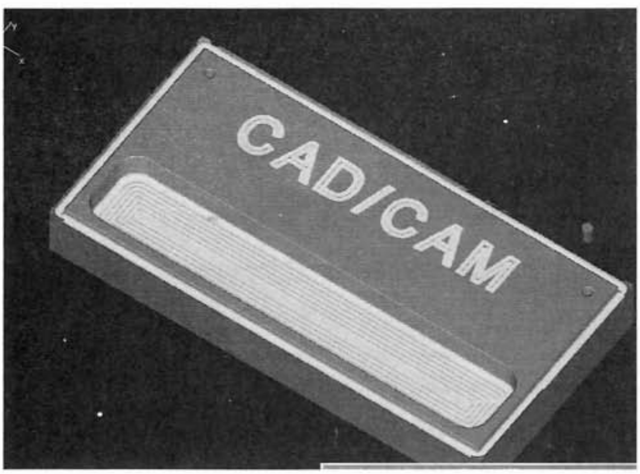
3. Select the **Use Job Setup values** button in the dialog box and enter the values below.



4. Choose **OK**.

5. Choose the **Machine** button from the Verify toolbar.

The part should look like the following picture.

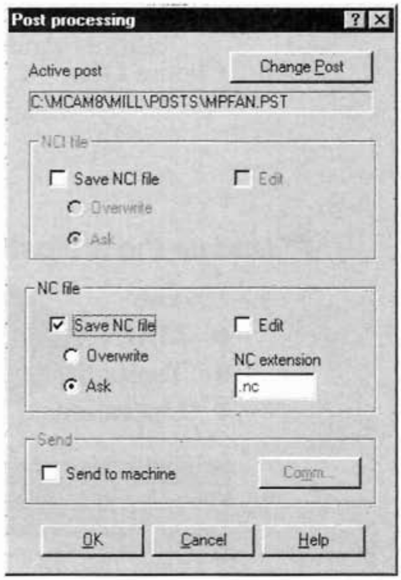


6. Close the Verify toolbar.

Post the NC Code

Note: Before continuing, ask your instructor if the correct post processor is set up in Mastercam. If it is not, skip to the next step and save the MC8 file.

1. Choose **Post**.



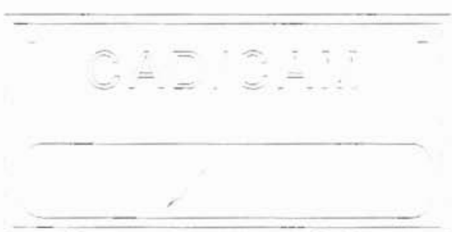
2. Enter the values shown in the dialog box at right.
3. Choose
   - **OK**
   - **Save**
4. Choose **OK** to close the Operations Manager.

Save the MC8 file again

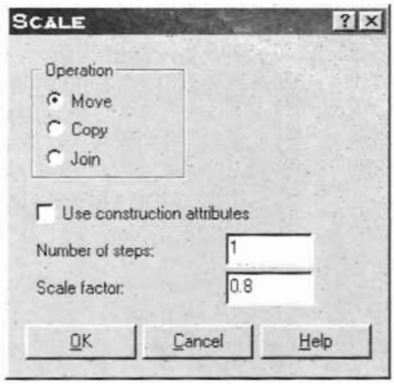
1. Choose
   - **MAIN MENU**
   - **File**
   - **Save**
   - **Save**
   - **Yes**

Associativity – Modify the existing geometry

1. Choose
   - **MAIN MENU**
   - **Xform**
   - **Scale**
   - **Chain**
2. Select a point on the pocket to begin chaining.

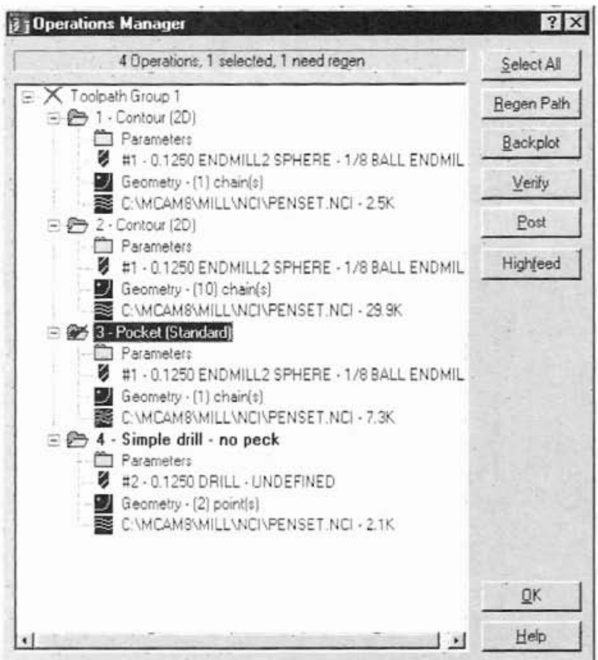


3. Choose
   - **Done**
   - **Done**
   - **Origin**
4. Enter the values in the dialog box at right.
5. Choose **OK**.



▶ *Update the toolpath*

1. Choose
   - **MAIN MENU**
   - **Toolpaths**
   - **Operations**



2. Select the third toolpath in the Operations Manager.

Note: Notice the red x, indicating the path needs to be regenerated.

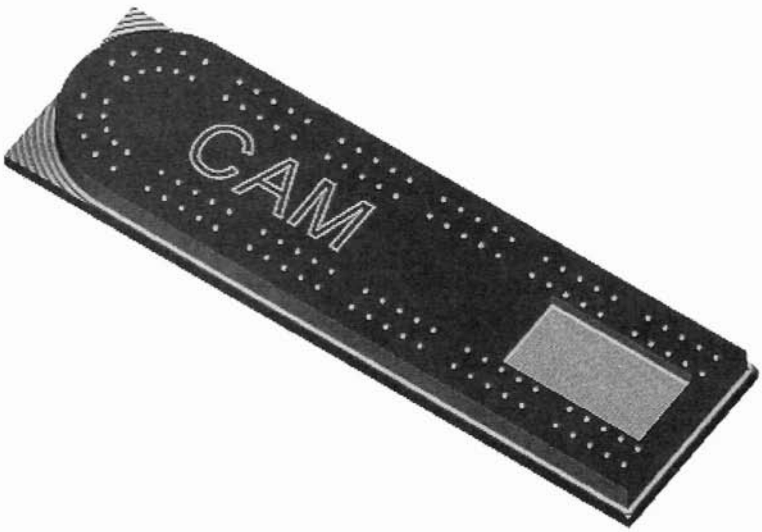
3. Choose
   - ◆ **Regen Path.**
   - ◆ **Select All**
4. Choose **Verify**.

5. Choose the **Machine** button from the Verify toolbar.

The part should look like the following picture.

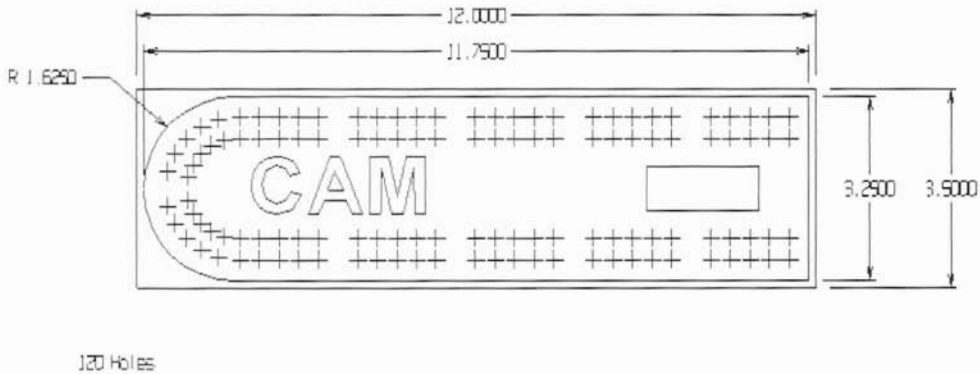
6. Save the file.

6 Cribbage Board



Objectives

- Navigate the Mastercam menus.
- Create lines, arcs, points and letters.
- Translate and offset lines.
- Enter coordinates.
- Construct 3D geometry.
- Use chaining.
- Create contour, drill, and pocket toolpaths.
- Enter parameters for contour, drill and pocket toolpaths.
- Perform solid model verification of the toolpath.



Geometry creation

▶ *Create a rectangle*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Rectangle**
   - ◆ **2 points**
2. *Enter the coordinates.* **X0, Y0**
3. *Enter the coordinates.* **X12, Y3.5**

4. Choose the **Fit** button on the toolbar to center the geometry.

Create the offset lines and corner fillets

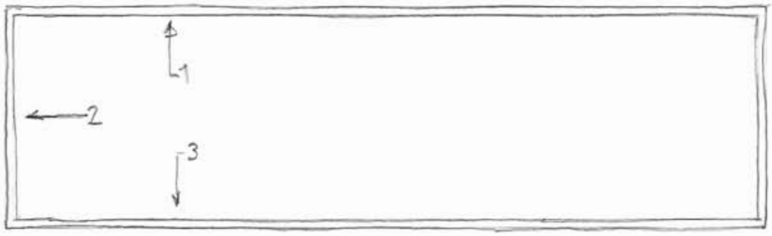
1. Choose
   - **MAIN MENU**
   - **Xform**
   - **Ofs ctour**
2. Select the line on the rectangle as shown on the following picture.

3. Choose **Done**.
4. Enter the values as shown in the dialog box at right.
5. Choose **OK**.

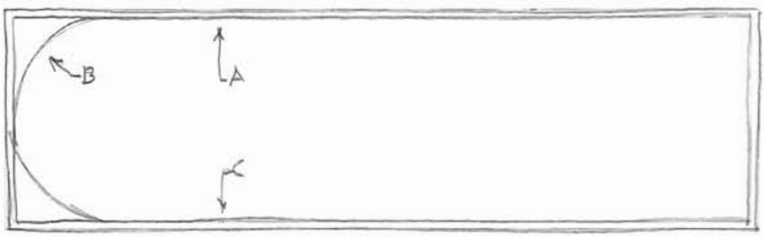
A smaller rectangle should appear inside the larger rectangle as shown in the following picture.

6. Choose
   - **MAIN MENU**
   - **Create**
   - **Arc**
   - **Tangent**
   - **3 ents/pts**

7. Select points **1, 2** and **3** on the inside of the rectangle as shown in the following picture.



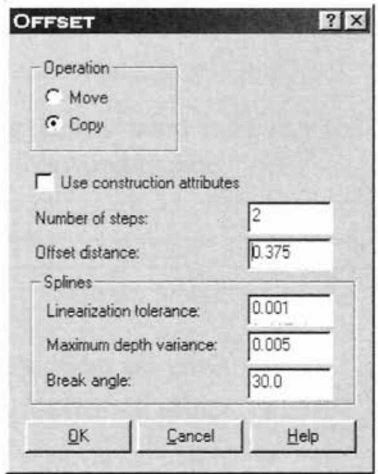
8. Choose
   - **MAIN MENU**
   - **Modify**
   - **Trim**
   - **2 entities**



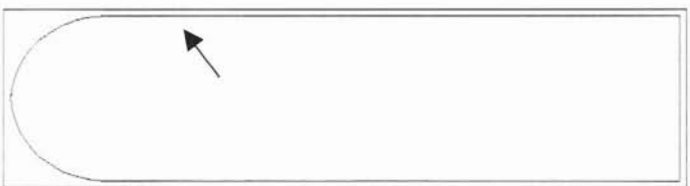
9. Select **A** as the line to trim.
10. Select **B** as the arc to trim to.
11. Select **C** as the line to trim.
12. Select **B** as the arc to trim to.
13. Choose
    - **MAIN MENU**
    - **Delete**
14. Choose the inside left vertical line to delete.

The part should look like the following picture.

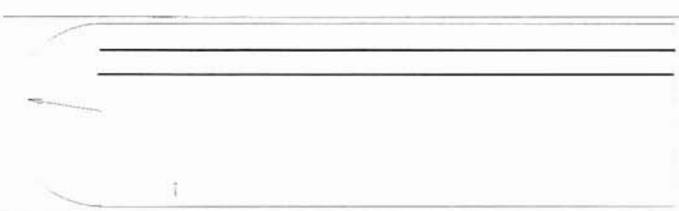
▶ *Offset entities*



1. Choose
   - **MAIN MENU**
   - **Xform**
   - **Offset**
2. Enter the values shown in the dialog box at right, including entering 2 for the Number of steps.
3. Choose **OK**.
4. With the cursor, click on the line as shown in the diagram below, bring the mouse toward the center of the rectangle and click again.

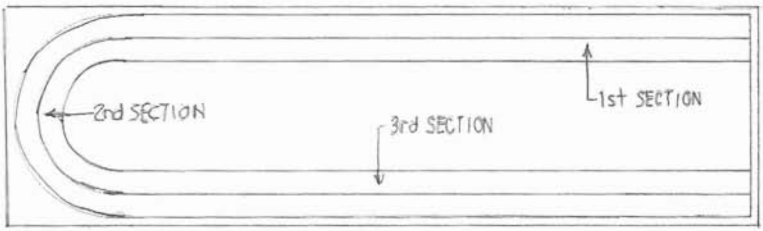


5. Repeat this on the arc and the bottom line of the same contour, bringing your mouse to the center and clicking after each selection.



▶ *Create points for peg holes*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Point**
   - ◆ **Along ent**
2. Select the first section of the middle contour.

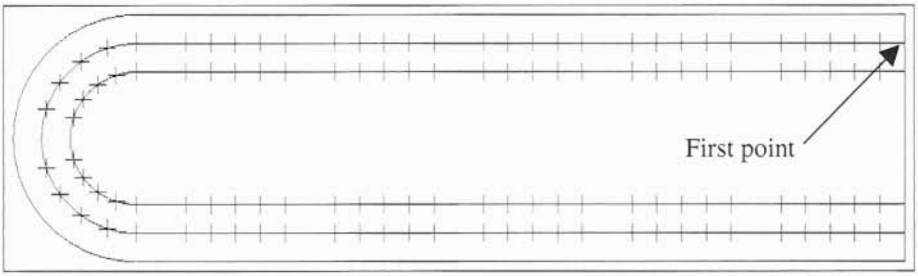


3. *Enter the number of points to create.* **32**
4. Select the second section (arc) of the middle contour.
5. *Enter the number of points to create.* **11**
6. Select the third section of the middle contour.
7. *Enter the number of points to create.* **32**
8. Repeat for each section of the inner contour.
9. Choose **Color** from the Secondary Menu.
10. Enter #13 (magenta), or select the color square and choose **OK**.
11. Choose
    - ◆ **MAIN MENU**
    - ◆ **Screen**
    - ◆ **Chg colors**
    - ◆ **All**
    - ◆ **Points**
    - ◆ **Done**
12. Choose
    - ◆ **MAIN MENU**
    - ◆ **Delete**

- **Duplicate**
- **Points**

Note: The prompt at the bottom of the screen shows 4 duplicate points have been discarded.

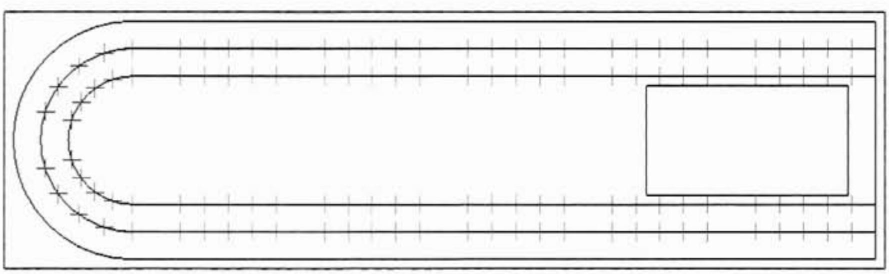
13. Press [Enter].
14. Choose **Repaint** from the toolbar.
15. Choose
    - **Only**
    - **Points**
16. Delete unnecessary points by selecting the first point and every sixth point. The result should look like the following picture.



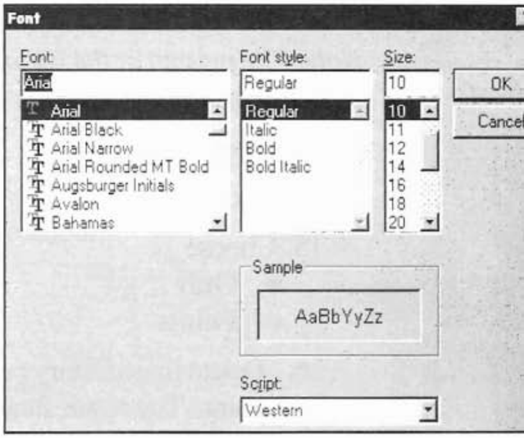
▶Create a rectangle

1. Choose
    - **MAIN MENU**
    - **Create**
    - **Rectangle**
    - **2 points**
2. *Enter coordinates.* **8.75, 1.125**
3. *Enter coordinates.* **11.5, 2.375**

The result should look like the following picture with 120 points.

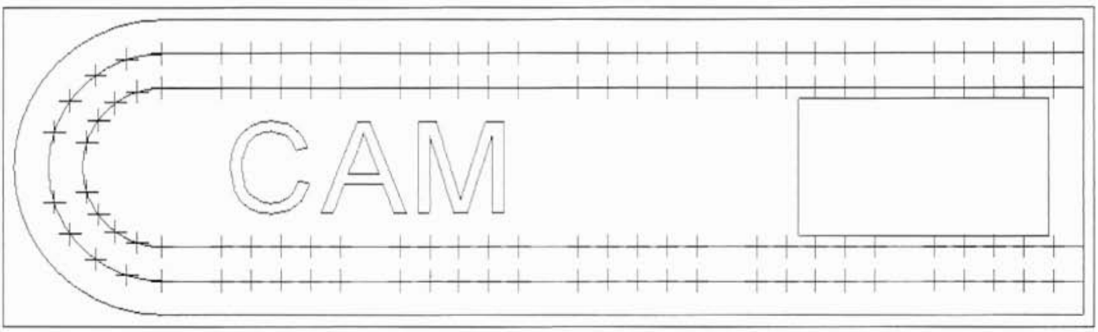


Create the letters CAM



1. Choose
   - **BACKUP**
   - **BACKUP**
   - **Next menu**
   - **Letters**
   - **True Type®**
2. Enter the parameters shown at right.
3. Choose **OK**.
4. *Enter the capital letters.* **CAM**
5. *Enter the height.* **1**
6. Choose **Horizontal**.
7. *Enter the spacing* **.125**
8. *Enter the starting location of the letters.* **2.5, 1.25**

The part should look like the following picture.



Save the drawing

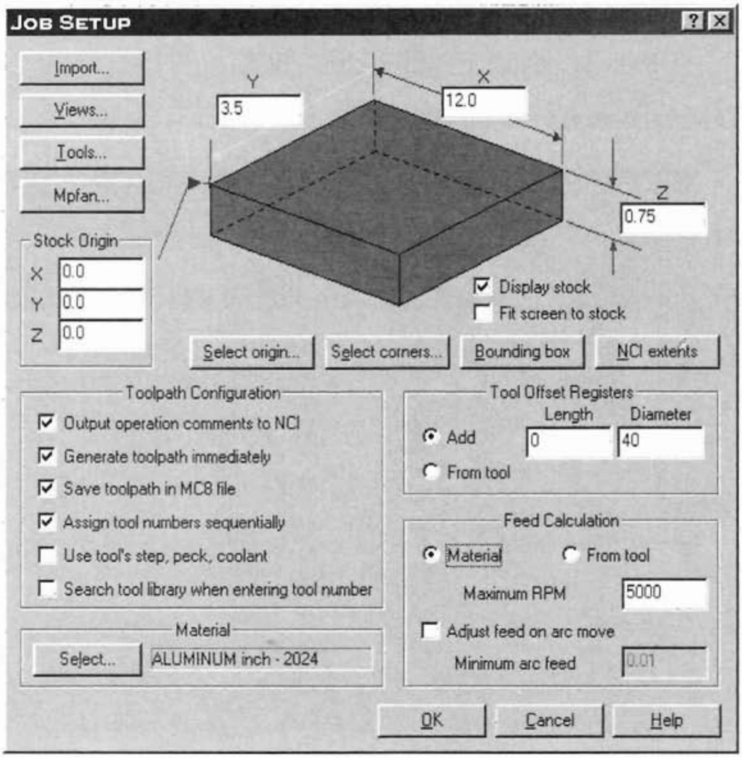
1. Choose
   - **MAIN MENU**
   - **File**
   - **Save**

2. Type the first six letters of your last name followed by the number 6.
3. Choose **Save**.

Toolpath creation

▶ *Define the rough stock*

1. Choose
   - **MAIN MENU**
   - **Toolpaths**
   - **Job setup**
2. Enter the values in the following dialog box.



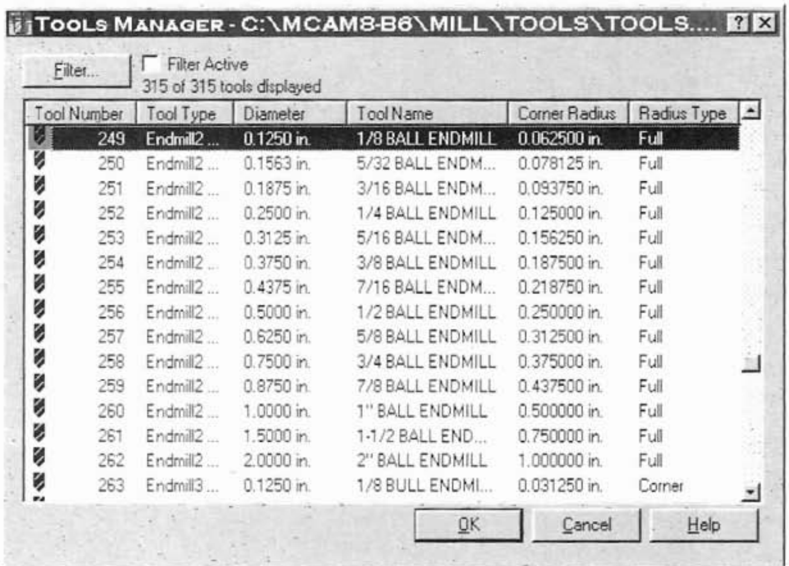
3. Move the red arrow to the left corner.
4. Choose **OK**.

Drilling the outside row holes

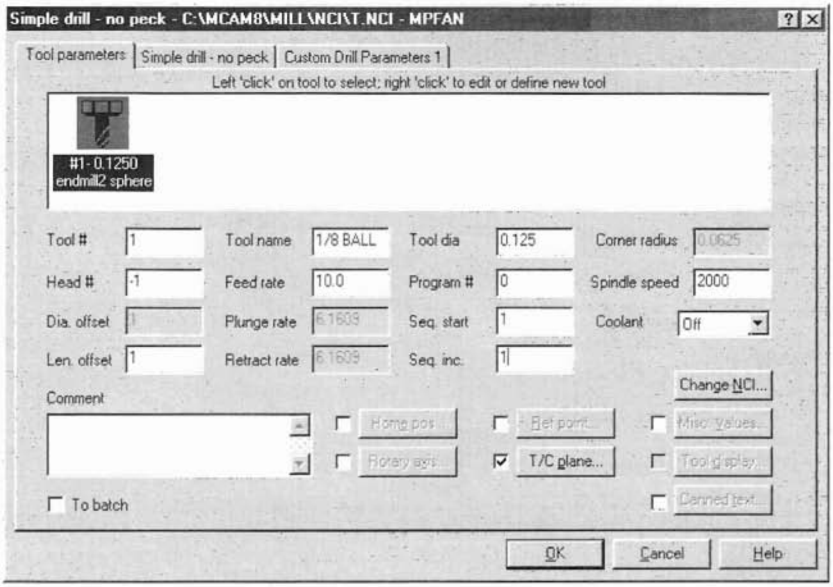
1. Choose
   - **Drill**
   - **Entities**
   - **All**
   - **Points**
   - **Done**
   - **Done**

Drilling the parameters

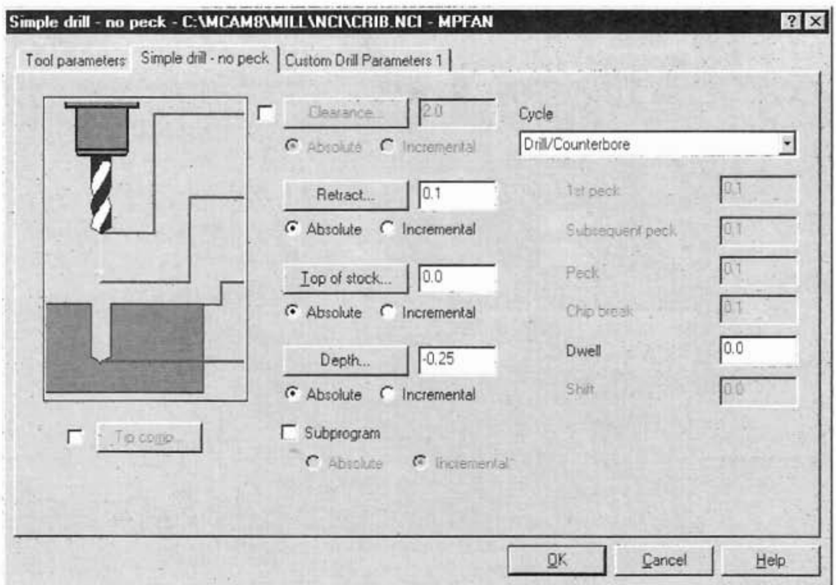
1. Right-click in the tool display area.
2. Choose **Get tool from library**.
3. Deselect the Filter Active check box.
4. Select a 1/8" ball endmill.



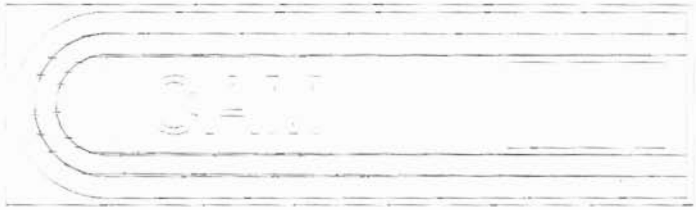
5. Choose **OK**.
6. Enter the values shown in the following dialog box.



7. Select the **Simple drill-no peck** tab and enter the values shown.

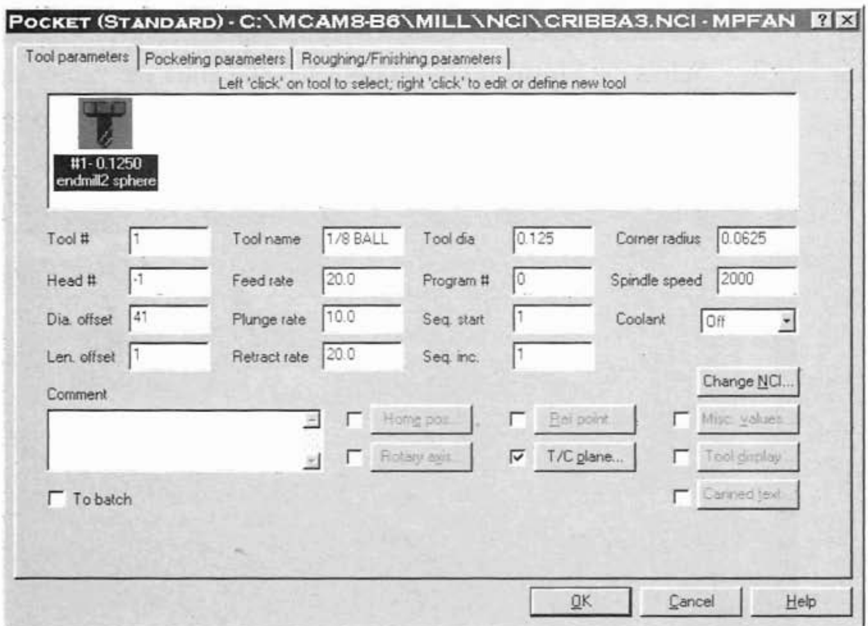


8. Choose **OK**. The toolpath should look like the following picture.

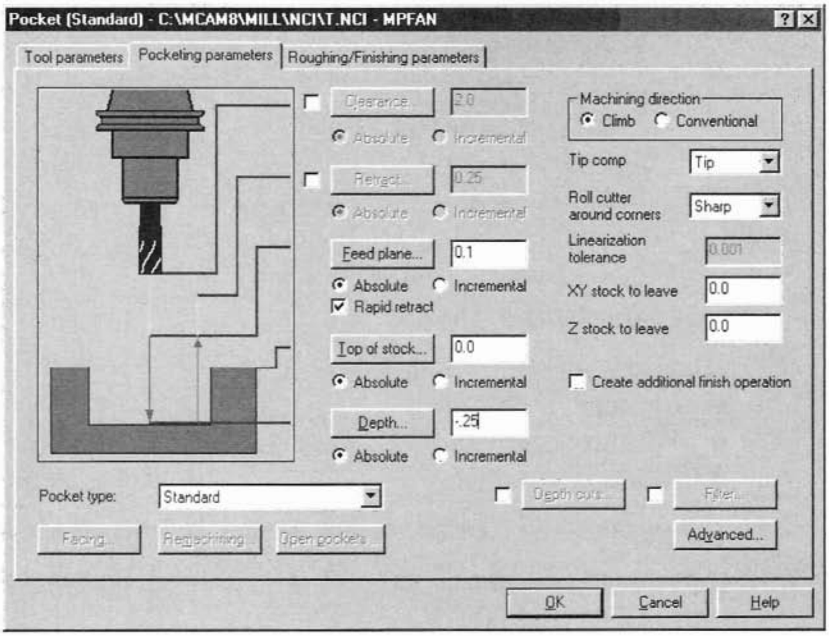


▶ *Program the pocket*

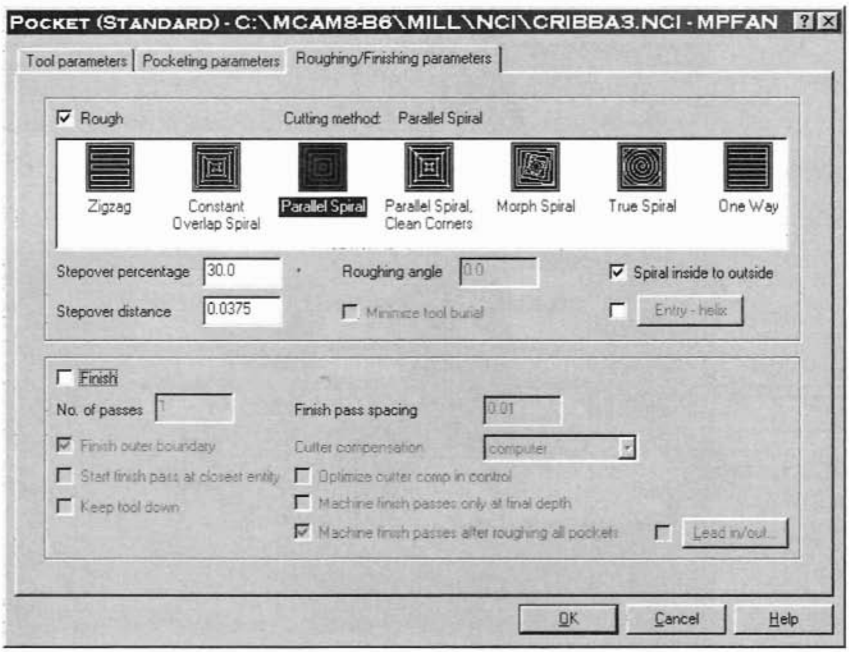
1. Choose **Pocket**.
2. Select a line on the small rectangle.
3. Choose **Done**.
4. Enter the values shown in the following dialog box.



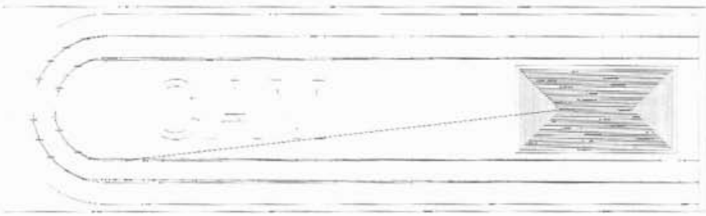
5. Select the **Pocketing parameters** tab and enter the values shown on the following dialog box.



6. Select the **Roughing/Finishing parameters** tab and enter the values shown.



7. Choose **OK**. The toolpath should look like the following picture.

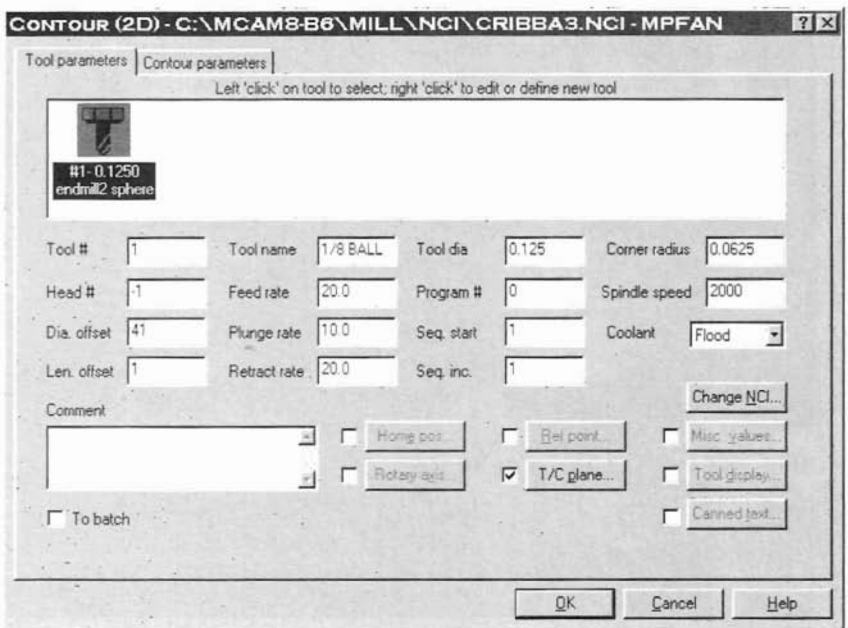


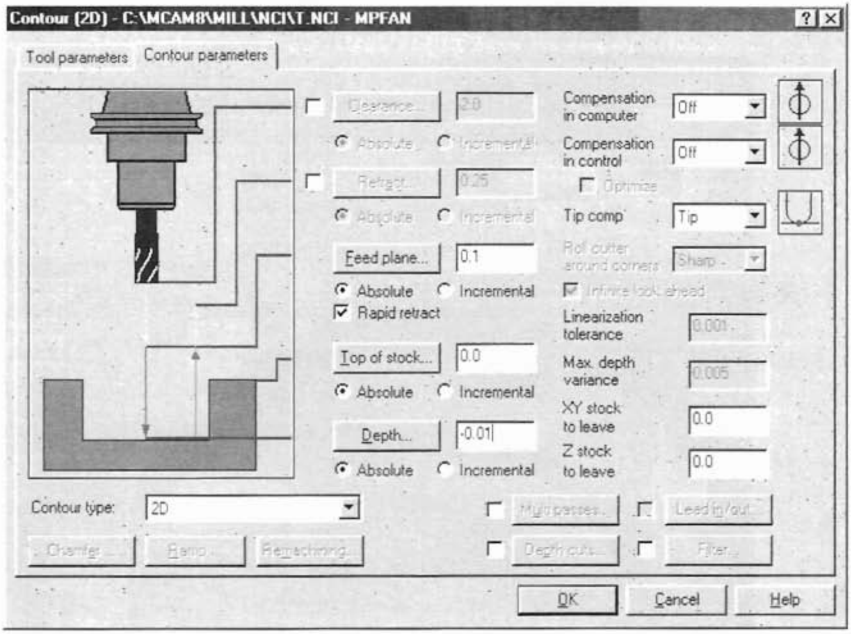
▶ *Engrave the letters*

1. Choose
   - **Contour**
   - **Window**
   - **Inside**
2. Using the cursor, draw a window around the letters **CAM**.

Note: Make sure the window includes all the letters, but not other lines.

3. Select a point inside the letter **C**.
4. Choose **Done**.
5. Enter the values shown on the following dialog boxes.



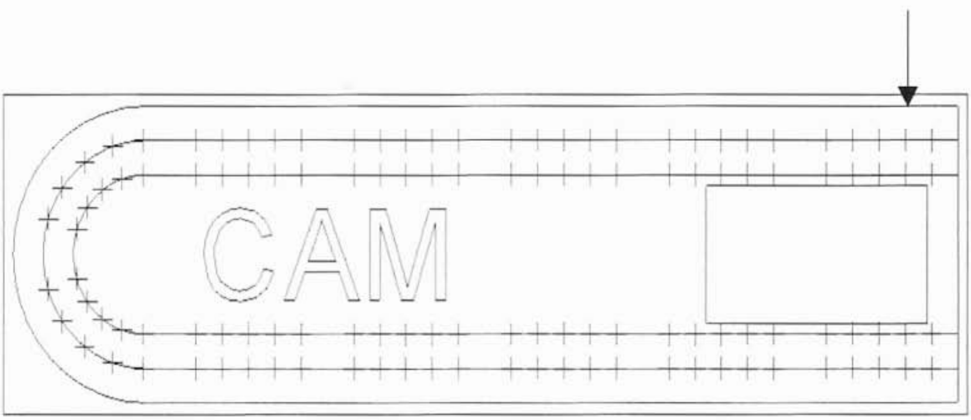


Note: Make sure Compensation in computer is set to Off.

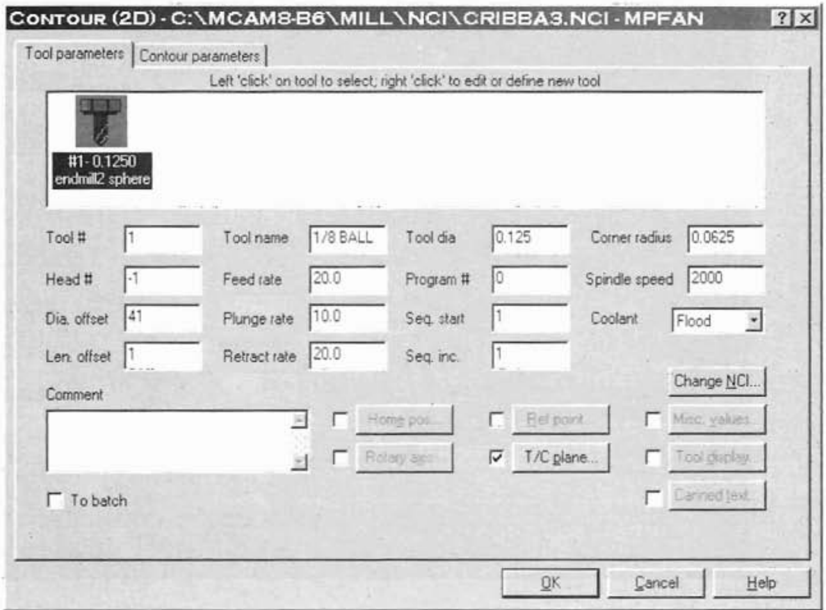
6. Choose **OK**.

▶ *Machine the outside contour*

1. Choose **Contour**.
2. Select the top right corner of the contour as shown in the following picture.
3. Choose **Done**.

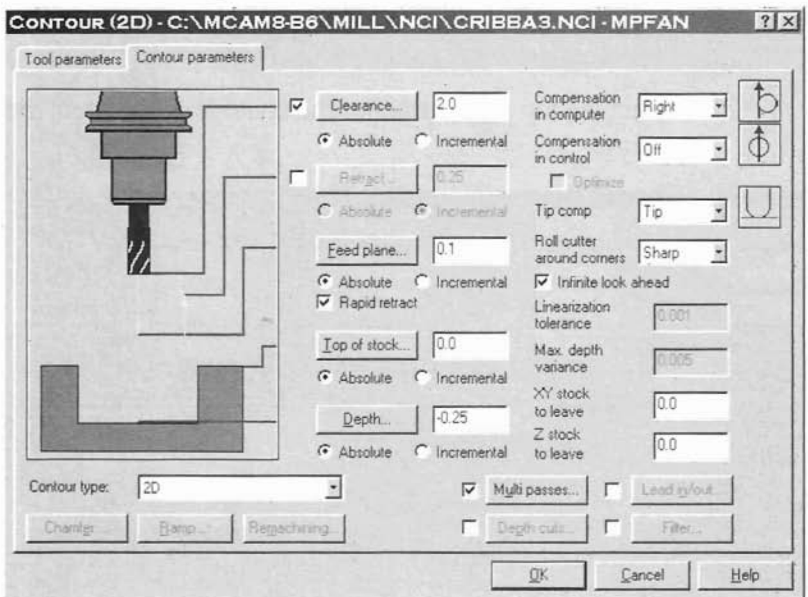


4. Enter the values shown on the following dialog box.



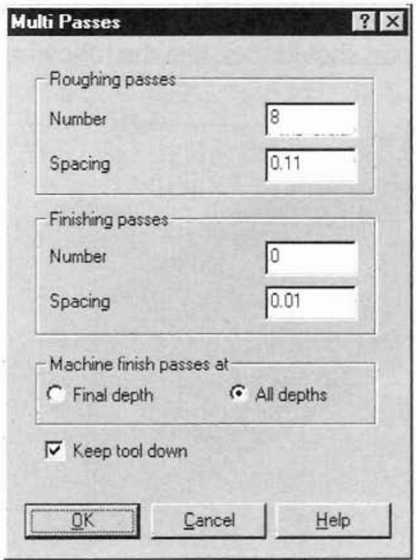
5. Enter the values shown in the following dialog box.

6. Select the **Multi passes** check box.



7. Choose the **Multi passes** button.

8. Enter the values shown in the following dialog box.



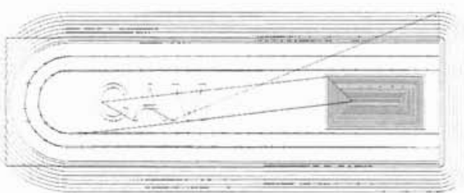
9. Choose **OK**.
10. Choose **OK**.

Check the program by backplotting

1. Press [Alt+O] to open the Operations Manager.
2. Choose
   - **Select All**
   - **Backplot**
   - **Run**

Note: Try checking the toolpath in a number of different views or angles.

3. The toolpath should look like the following picture.

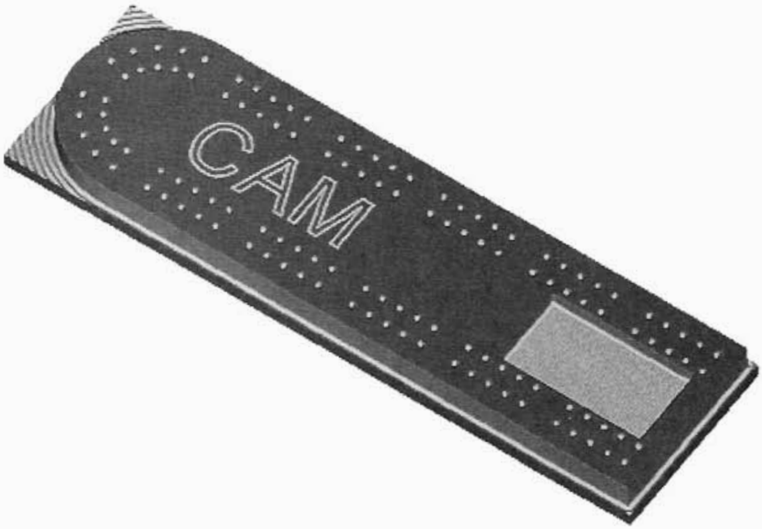


4. Press [Esc] to return to the Operations Manager.

▶ Verify the program

Refer to verifying in the previous chapter if you need to.

The verified part should look like the following picture.



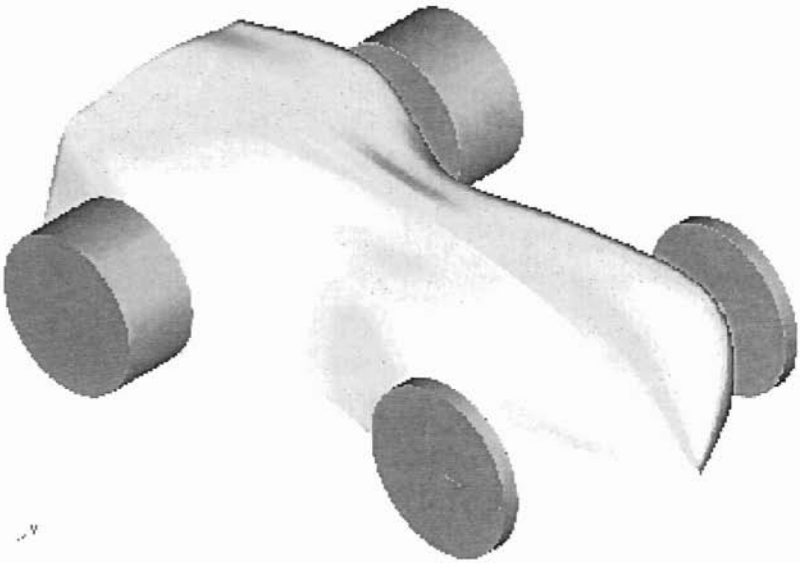
▶ Post the NC Code

Refer to posting in the previous chapter if you need to.

▶ Save the MC8 file

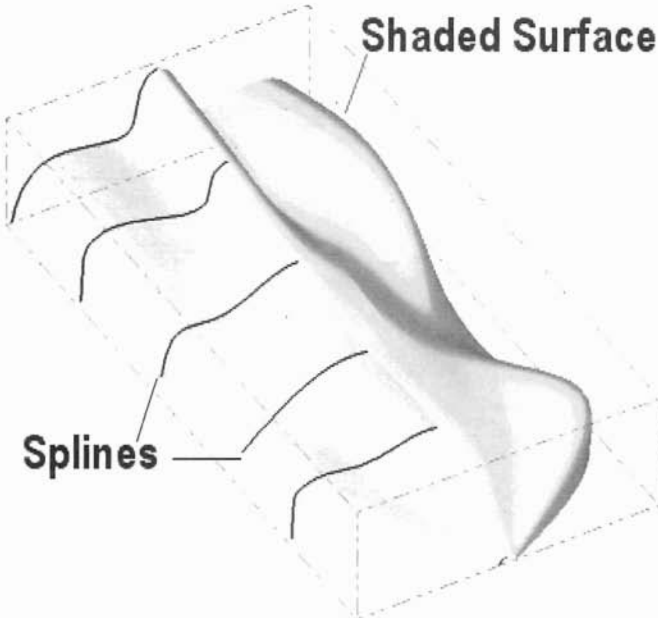
Refer to saving in the previous chapter if you need to.

7 Gravity Car



Objectives

- Create splines and 3D surfaces.
- Manipulate shading and surfaces.
- Construct 3D geometry.
- Select surfaces.
- Create flowline toolpaths.
- Enter parameters for a flowline toolpath.
- Perform solid model verification of the toolpath.



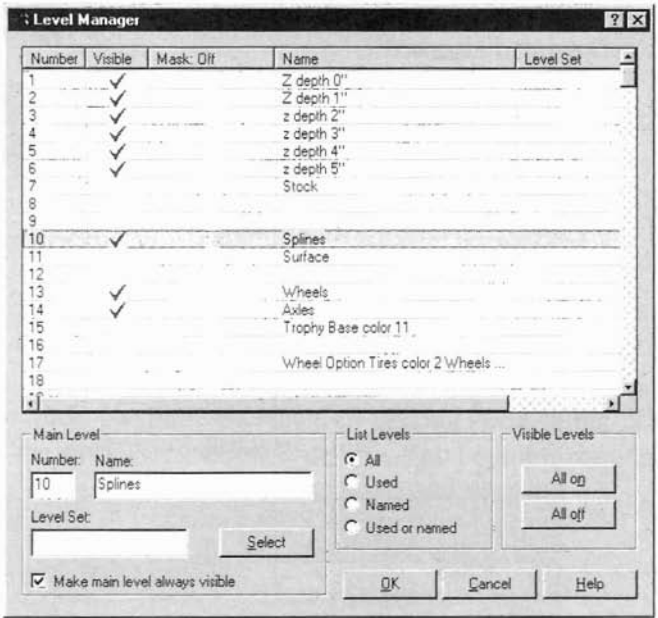
Geometry creation

▶ *Open the concept car template*

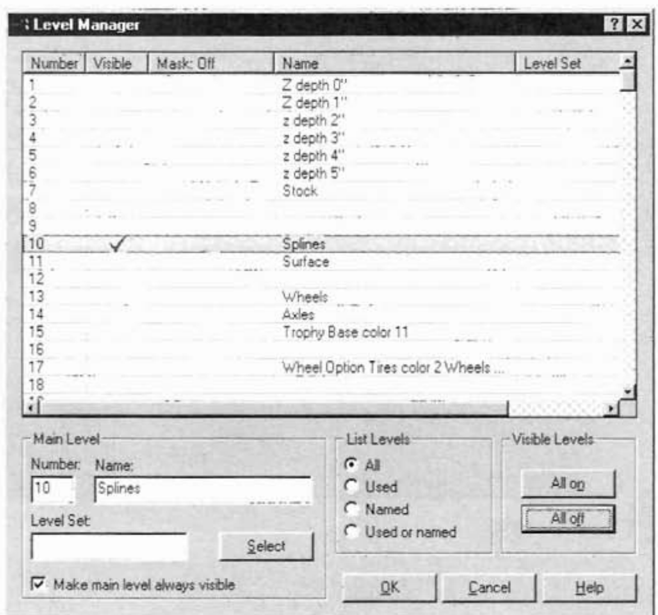
1. Choose
   - **MAIN MENU**
   - **File**
   - **Get**
2. Select the Cartemp.MC8 file from the Mill\MC8 directory or from the floppy disk as directed by your instructor.
3. Save the file using the first six letters of your last name followed by the letters "car".

▶ *Create splines*

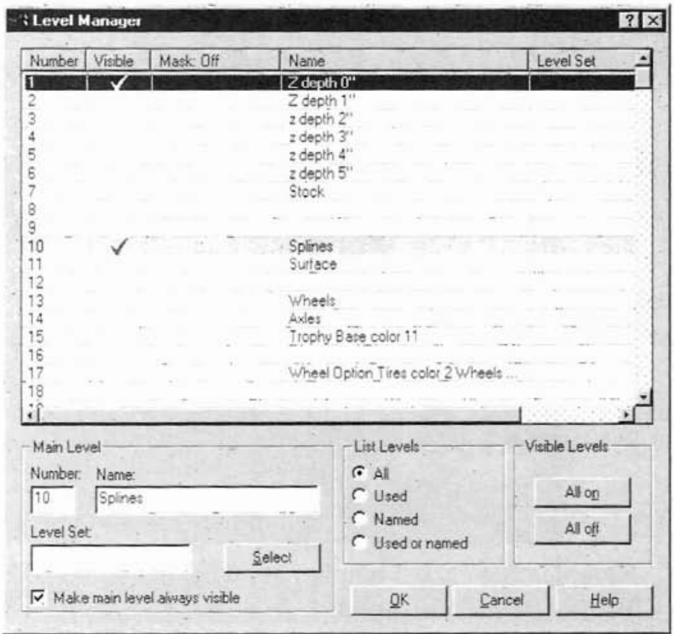
1. Choose **MAIN MENU**.
2. Choose **Level** from the Secondary Menu.



3. Double-click on the number **10** in the dialog box. The Splines line will be highlighted in yellow. This is now the active level.
4. Choose **All** for List Levels in the dialog box.
5. Choose **All off** for Visible Levels. The result should look like the following dialog box.



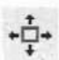
6. In the **Visible** column, select **Level 1, Z depth 0"** as shown in the following dialog box.



7. Choose **OK**.

8. Choose the **Gview (side)** green button from the toolbar.

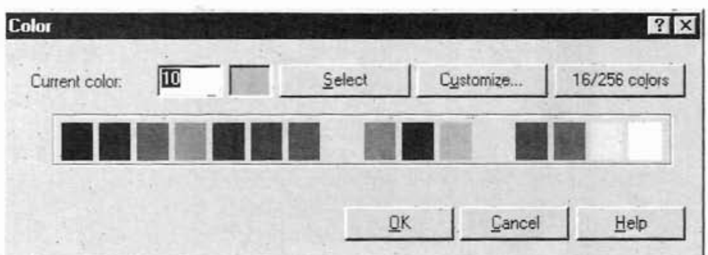
9. Choose the **Cplane (side)** blue button from the toolbar.



10. Choose the **Fit** button on the toolbar to center the geometry.

11. Choose **Color** from the Secondary Menu.

12. Enter **10** (green) as shown in the following dialog box.



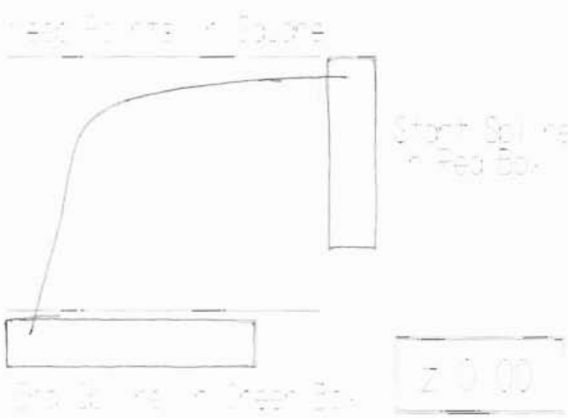
13. Choose **OK**.

14. Choose **Z** from the Secondary Menu.
15. Enter **0** as the new construction depth.
16. Right-click in the graphics window.
17. Deselect **AutoCursor** in the right-click menu.

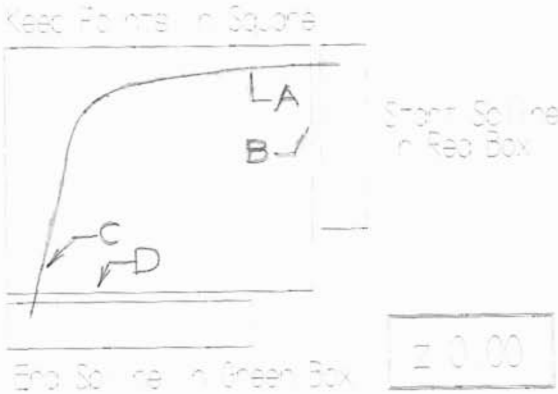
▶ *Create a spline*

You will define the shape of your car by creating 6 splines (cross sections). You will create the left side of the car and then mirror it to create the other side. Your visual orientation while creating splines will be looking from the front of the car to the back. A spline is created by placing node points. The fewer the points used, the smoother the spline will be. You begin by drawing the back spline.

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Spline**
   - ◆ **Manual**
2. Start the spline by clicking in the upper part of the red box and continue clicking points left and down until you end up in the green box. Each point should be to the left and below of the proceeding point as shown in the following picture. Your spline may not look exactly like the one below.
3. Press [Esc] to complete the spline. If you don't like the spline, you can delete it and create a new spline.
4. Choose
   - ◆ **MAIN MENU**
   - ◆ **Modify**
   - ◆ **Trim**
   - ◆ **1 Entity**



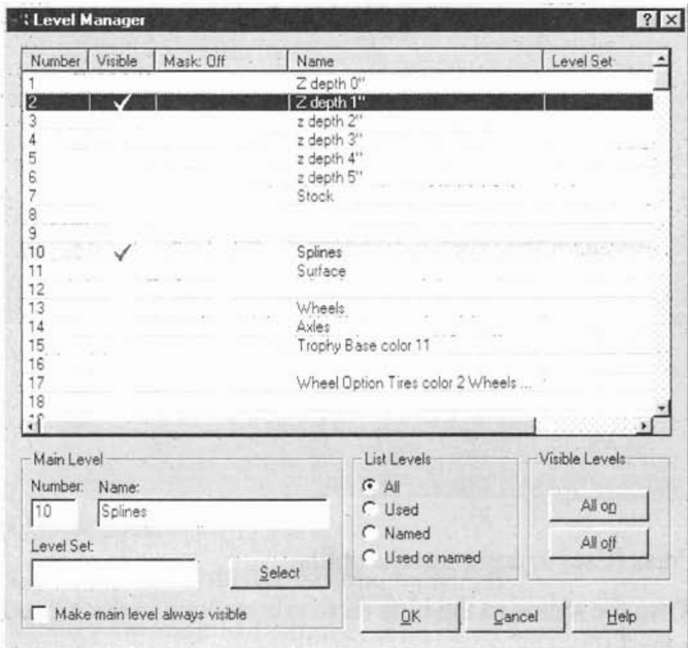
5. Select **A** as the part to keep.
6. Select **B** as the line to trim to.
7. Select **C** as the part to keep.
8. Select **D** as the remaining blue line to trim to.



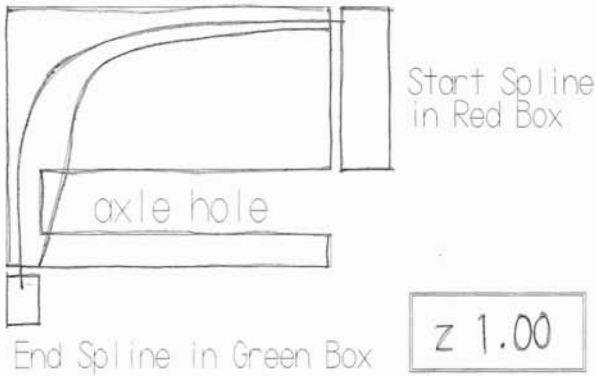
Create the second spline

1. Choose **Z** from the Secondary Menu.
2. Enter **1** as the new construction depth. Your next spline will be 1" forward.
3. Choose **Level** from the Secondary Menu.

4. In the Visible column, deselect Level 1, Z depth 0".
5. Select **Level 2**, **Z depth 1** as shown in the following dialog box. (Splines should remain checked.)



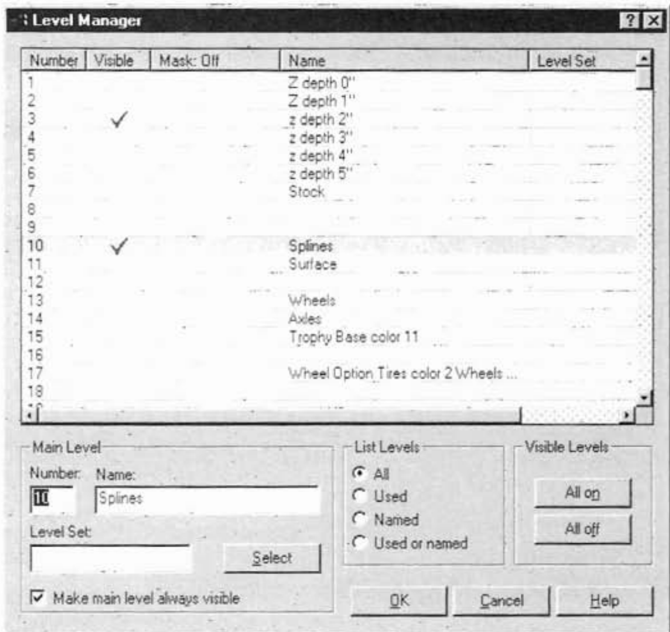
6. Choose **OK**.
7. Choose **Color** from the Secondary Menu.
8. Enter **11** (light blue).
9. Choose **OK**.
10. Choose
    - **MAIN MENU**
    - **Create**
    - **Spline**
    - **Manual**
    - **Sketch**
11. Start the spline by clicking in the red box and continue clicking points over and down until you end up in the green box. Each point should be to the left and below the proceeding point as shown in the following picture. The spline should remain outside the axle hole boundary and inside the blue lines.



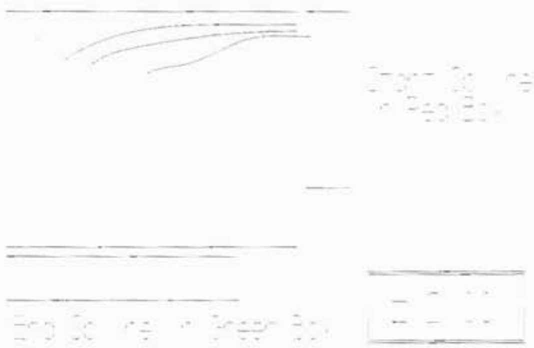
12. Press [Esc] to complete the spline.
13. Trim the spline to the blue rectangle as done in the preceding spline on page 110.

▶ *Create the third spline*

1. Choose **Z** from the Secondary Menu.
2. Enter **2** as the new construction depth.
3. Choose **Level** from the Secondary Menu.
4. In the Visible column, deselect Level 2, Z depth 1.
5. Select **Level 3, Z depth 2"** as shown in the following dialog box. (Splines should remain checked.)



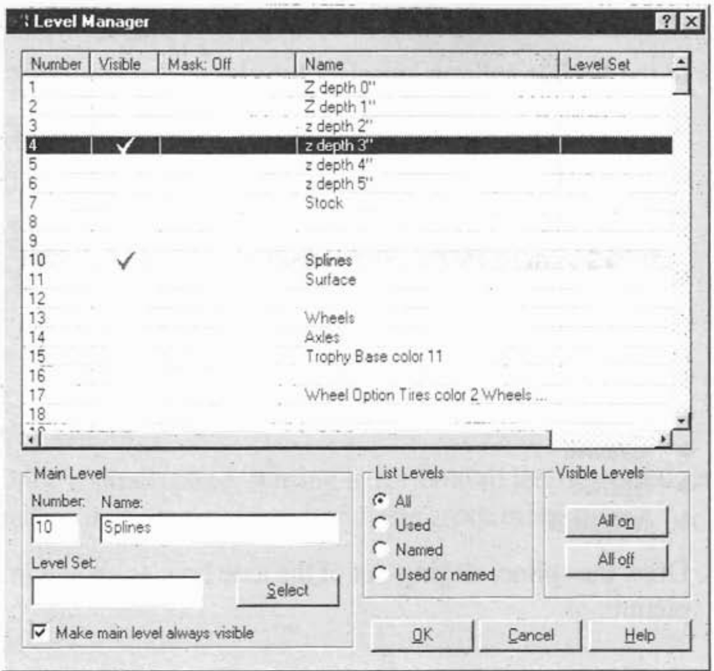
6. Choose **OK**.
7. Choose **Color** from the Secondary Menu.
8. Change the color to **12** (red).
9. Choose **OK**.
10. Choose
    - **MAIN MENU**
    - **Create**
    - **Spline**
    - **Manual**
11. Draw a spline, starting in the red box and clicking points until you end up in the green box as shown in the following picture.



12. Press [Esc] to complete the spline.
13. Trim the spline to the blue rectangle.

▶ *Create the fourth spline*

1. Choose **Z** from the Secondary Menu.
2. Enter **3** as the new construction depth.
3. Choose **Level** from the Secondary Menu.
4. In the Visible column, deselect Level 3, Z depth 2.
5. Select **Level 4, Z depth 3"** as shown in the following dialog box.

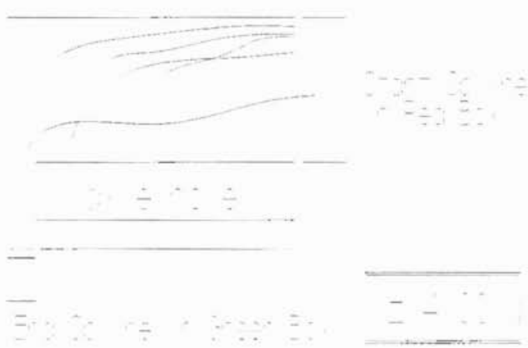


6. Choose **OK**.
7. Choose **Color**.
8. Enter **13** (purple).
9. Choose **OK**.
10. Choose
    - **MAIN MENU**
    - **Create**
    - **Spline**
    - **Manual**
11. Draw the spline starting in the top red rectangle and ending in the bottom green rectangle.
12. Press [Esc] to complete the spline.
13. Trim the spline.

▶ *Create the fifth spline*

1. Choose **Z** from the Secondary Menu.
2. Enter **4** as the new construction depth.

3. Choose **Level**.
4. In the Visible column, unselect Level 4, Z depth 3.
5. Select **Level 5, Z depth 4"**.
6. Choose **OK**.
7. Choose **Color**.
8. Enter **14** (yellow).
9. Choose **OK**.
10. Choose
    - **MAIN MENU**
    - **Create**
    - **Spline**
    - **Manual**
11. Draw the spline, staying out of the axle hole as shown in the following picture.

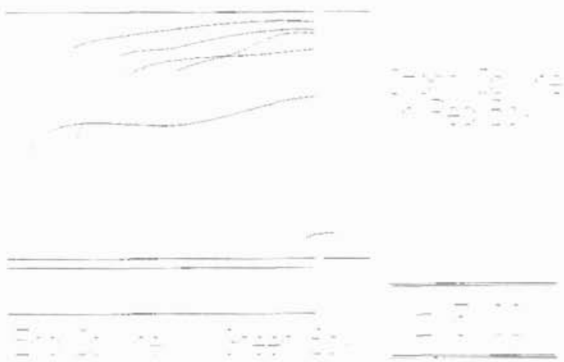


12. Press [Esc] to complete the spline.
13. Trim the spline.

Create the sixth spline

1. Choose **Z** from the Secondary Menu.
2. Enter **5** as the new construction depth.
3. Choose **Level**.
4. In the Visible column, deselect Level 5, Z depth 4.

5. Select **Level 6, Z depth 5"**.
6. Choose **OK**.
7. Choose **Color**.
8. Enter **15** (white).
9. Choose **OK**.
10. Choose
    - **MAIN MENU**
    - **Create**
    - **Spline**
    - **Manual**
11. Draw a small spline, starting in the bottom left of the red rectangle and ending in the top right corner of the green rectangle.



12. Press [Esc] to complete the spline.
13. Trim the spline.

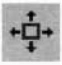
▶ *Create the surface*

1. Choose **Level**.
2. Double-click on the number 11 in the list area of the dialog box. The Surface line will be highlighted in yellow and a check mark will appear in the Visible column.
3. Choose **All off** for Visible Levels.
4. Select **Line 10 (Splines)** in the Visible column.

5. Choose **OK**.

6. Choose the **Gview (isometric)** button from the toolbar.

7. Choose the **Cplane (top)** button from the toolbar.



8. Choose the **Fit** button on the toolbar to center the geometry.

9. Choose **Color**.

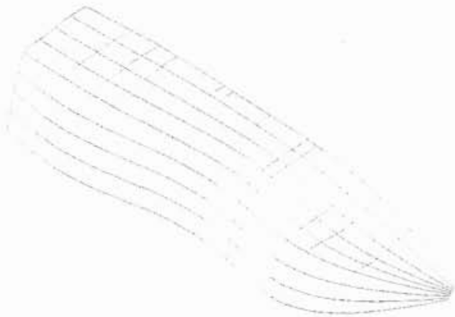
10. Enter **10** (green).

11. Choose
    - **MAIN MENU**
    - **Create**
    - **Surface**
    - **Loft**
    - **Single**

12. Select each spline in order and at the point specified in the following picture.

13. Choose
    - **Done**
    - **Do it**

A wireframe representation of the surface will appear, looking similar to the following drawing.



▶ *View the surface as a solid image*

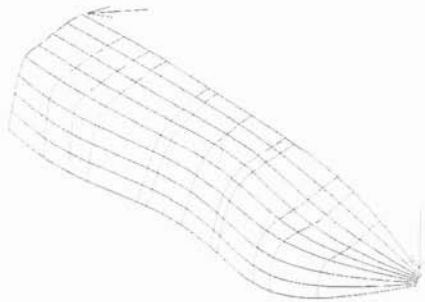
Shading allows you to view surfaces more easily.

1. Press [Alt+S] to shade the part or select the shading icon.
2. Turn the shading off by pressing [Alt+S] again.

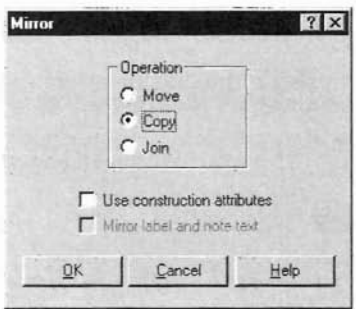
▶ *Mirror the surface*

1. Choose
   - **MAIN MENU**
   - **Xform**
   - **Mirror**
2. Select a line anywhere on the surface; it will highlight white.
3. Choose
   - **Done**
   - **2 points**
   - **Endpoint**

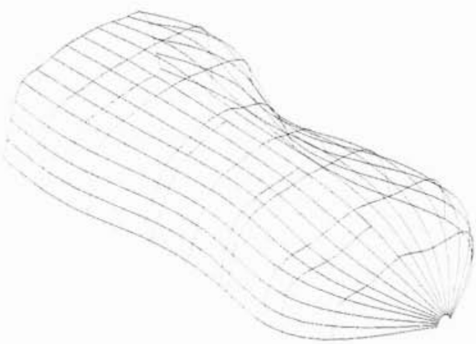
4. Select the points on the top of each end of the surface as shown in the diagram at right.



5. Choose **Copy** as shown in the following dialog box.



6. Choose **OK**. The surface is mirrored and should resemble the following picture.



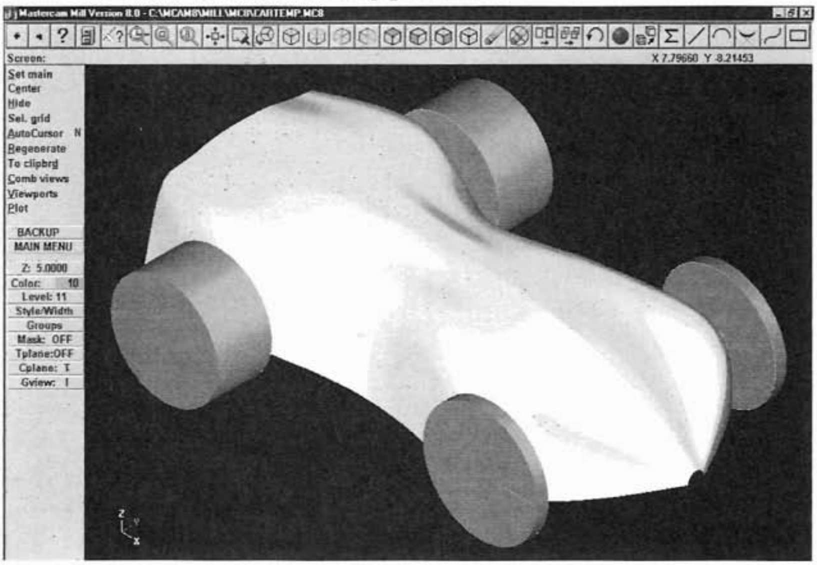
7. Fit the geometry to the screen.

8. Choose
   - **MAIN MENU**
   - **Screen**
   - **Clr colors**

9. Check the shape in different views using the dynamic icon.  Click in the middle of the car and drag the cursor to dynamically rotate the part.

▶ *Check the shape*

1. Choose **Level**.
2. Select the **Wheels level** in the Visible column.
3. Select the **Axles level** in the Visible column.
4. Deselect the **Splines level**.
5. Choose **OK**.
6. Fit the geometry to the screen.
7. Press [Alt-S] to shade the part and view it with wheels. It should look similar to the following picture.

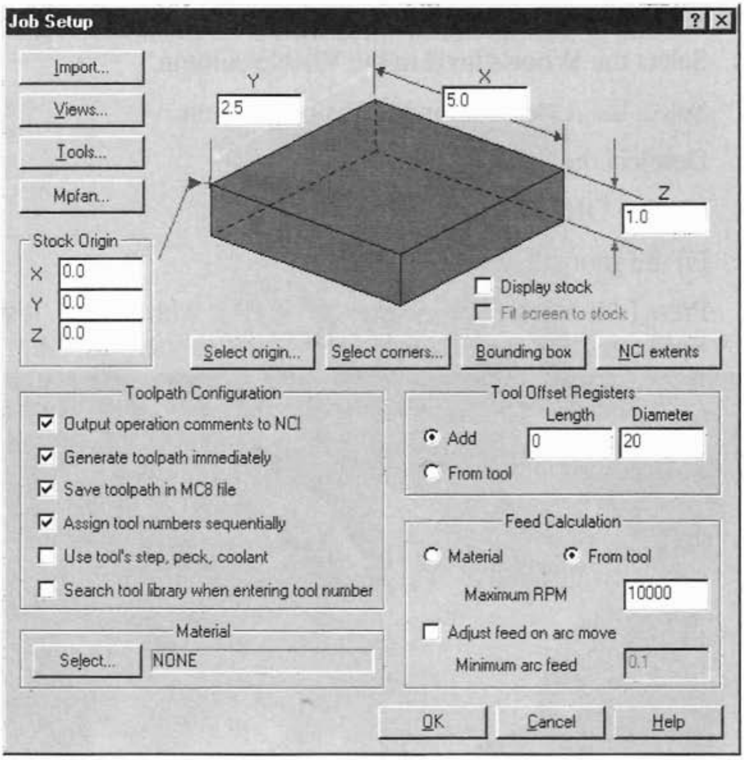


8. Press [Alt-S] to remove shading.
9. Choose:

- **MAIN MENU**
- **File**
- **Save**

Define the rough stock

1. Choose
   - **MAIN MENU**
   - **Toolpaths**
   - **Job Setup**
2. Enter the values shown on the following dialog box.

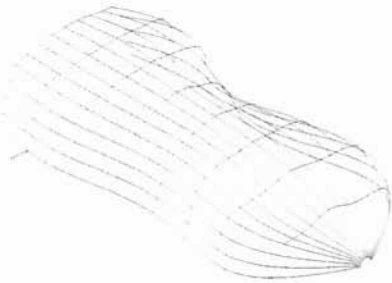


3. Choose **OK**.

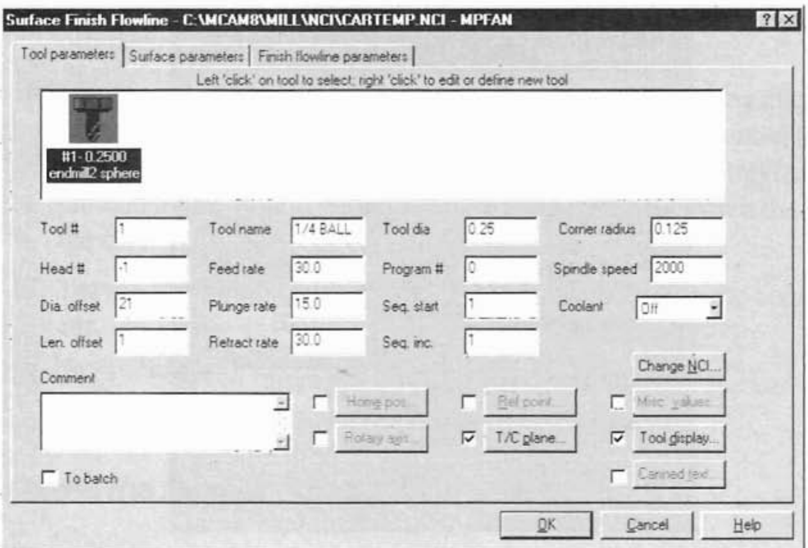
Create the first flowline toolpath

1. Choose **Level.**
2. Deselect all the levels except for Surface.
3. Choose **OK**.

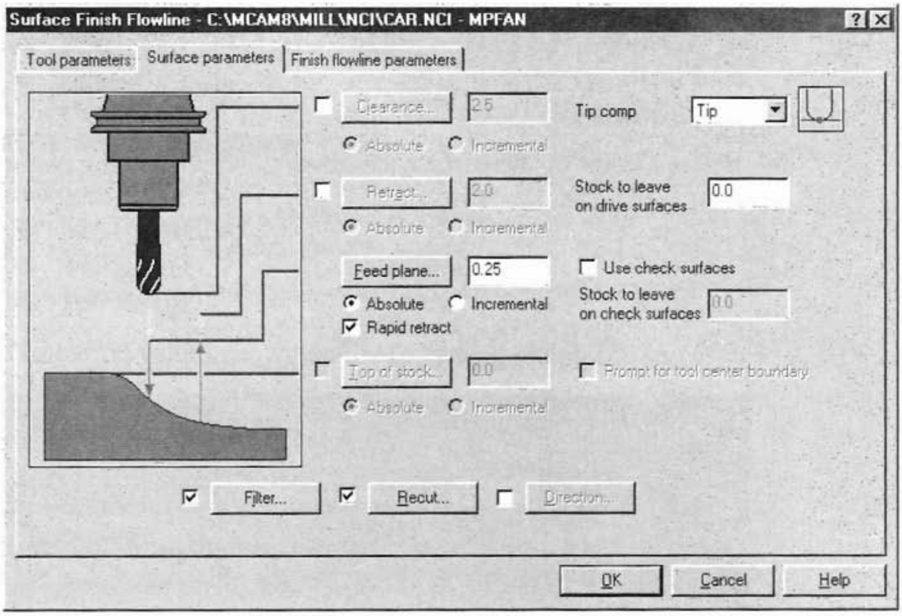
4. Choose
   - **Surface**
   - **Finish**
   - **Flowline**
5. Select a point on the surface as shown on the following picture.



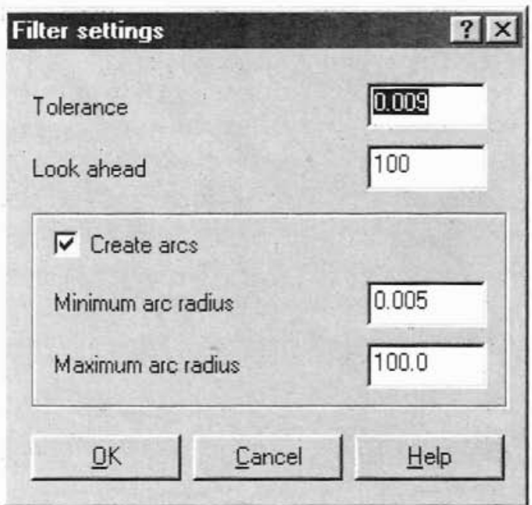
6. Choose **Done**.
7. Right-click in the tool display area and choose the ¼" ball endmill.
8. Enter the values shown on the following dialog box.



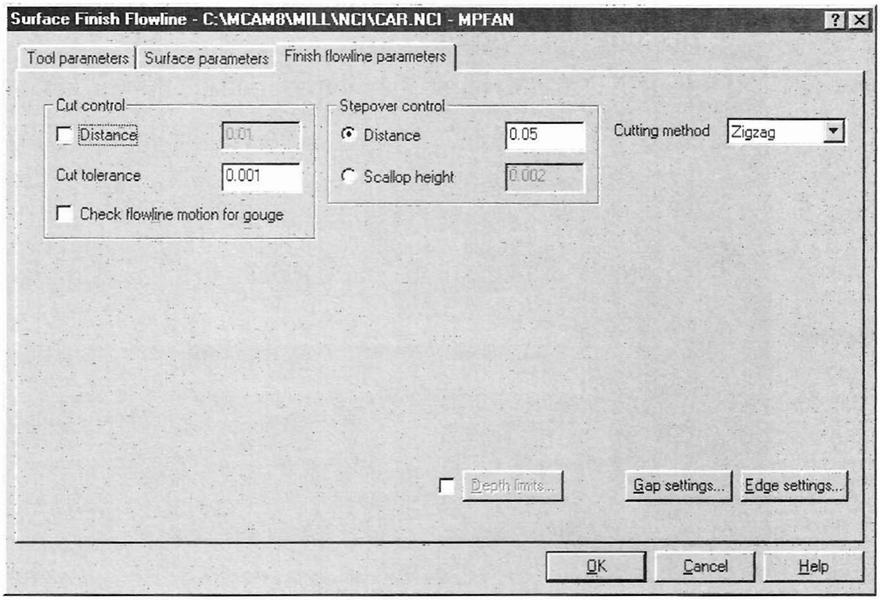
9. Choose the **Surface parameters** tab.
10. Enter the values shown on the following dialog box.



11. Select the **Filter** button and enter the values shown on the following dialog box.



12. Select the **Finish flowline parameters** tab.
13. Enter the values shown on the following dialog box.

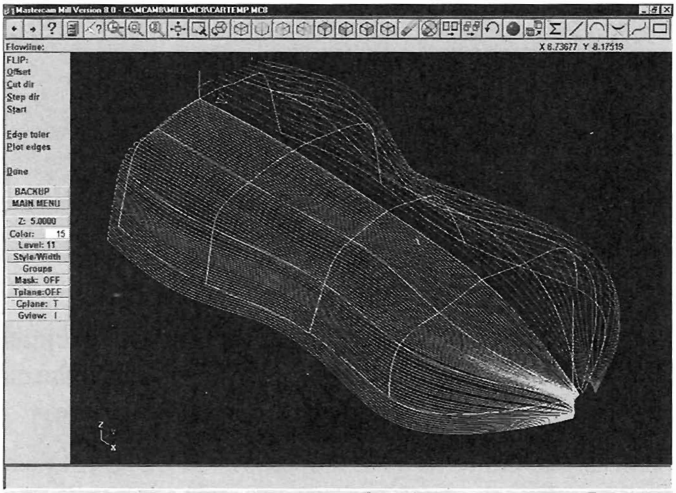


14. Choose **OK**.
15. Toggle **Offset** by clicking it with your mouse several times. You'll notice that the toolpath is either inside or outside the car. Toggle it so the offset display line is outside the car.
16. Just as you did for Offset, toggle **Cut dir** several times by clicking your mouse. This time, you'll notice the toolpath is either parallel to length of the car or perpendicular (side to side) to the length. Toggle it so it is parallel to the length, with the larger arrow pointing down the length of the car.
17. Toggle **Step dir** to move the toolpath starting point to the center of the car.
18. Toggle **Start** to move the starting point to the back of the car.
19. Choose **Done**.

Create the second flowline toolpaths

1. Choose
   - **Finish**
   - **Flowline**
2. Select the surface on the opposite side of the car.

3. Enter the same values in the next three dialog boxes as you did for the first flowline toolpath.
4. Toggle **Offset** to move the toolpath outside the car.
5. Toggle **Cut dir** to change the cutting direction to lengthwise.
6. Toggle **Step dir** to change the toolpath starting point to the top of the car.
7. Toggle **Start** to move the starting point to the back of the car.



8. Choose **Done**.

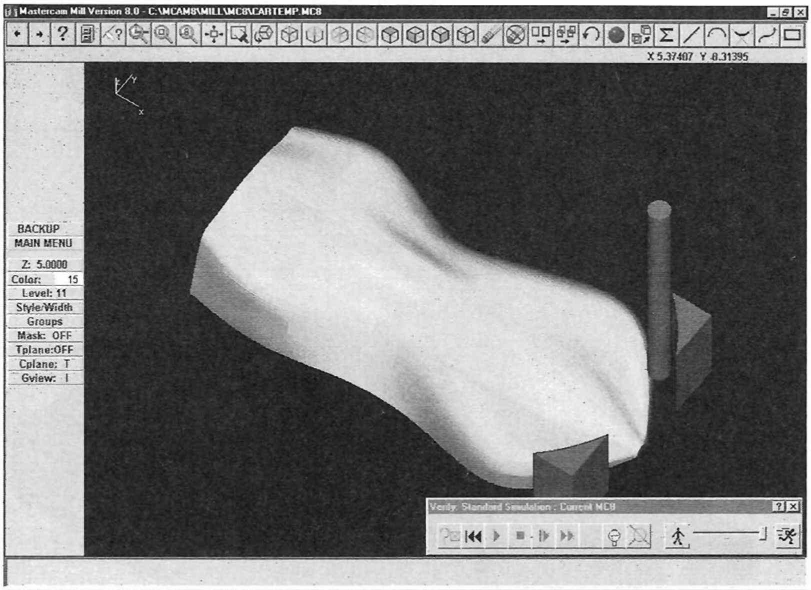
▶ *Checking the toolpath*

1. Choose **Level**.
2. Deselect all levels except Stock and Surface.
3. Choose **OK**.
4. Choose
   - **MAIN MENU**
   - **Toolpaths**
   - **Operations**
   - **Select All**

▶ *Verify*

Refer to the previous chapter if you need help.

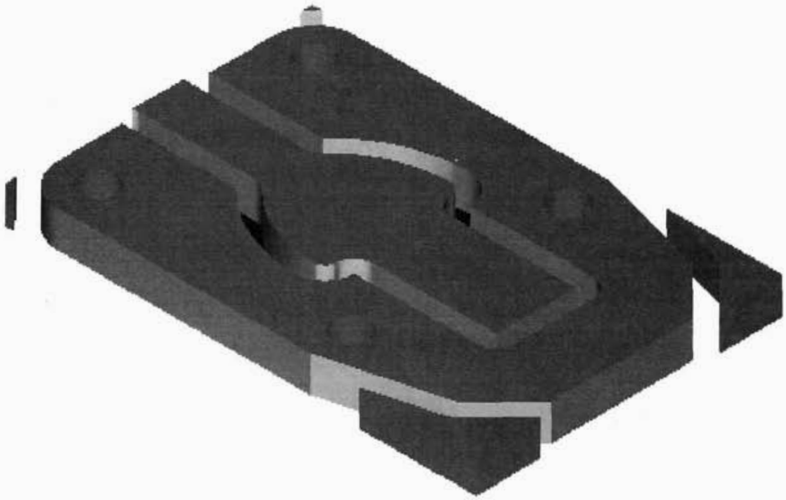
Your part should resemble the following picture.



▶ *Save*

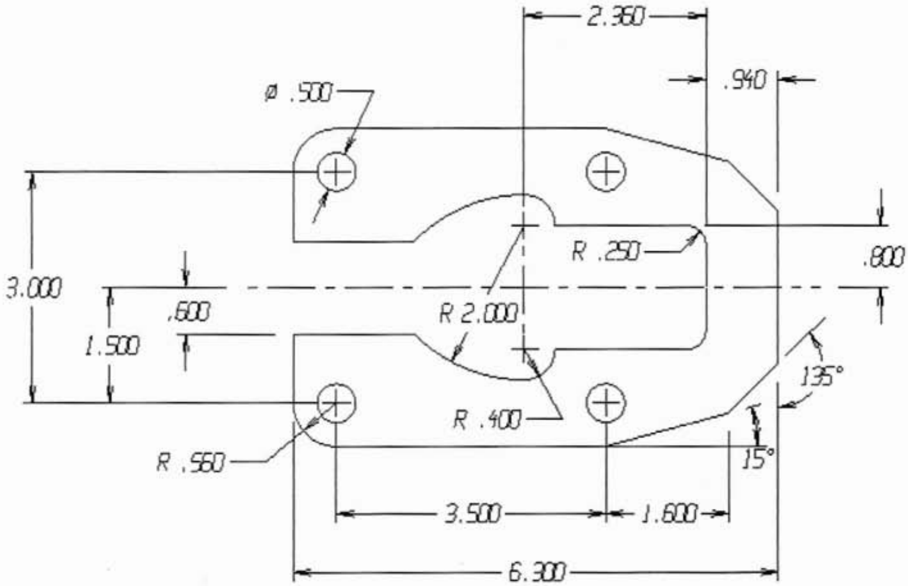
Refer to previous chapters if needed.

8 2D Drill and Contour



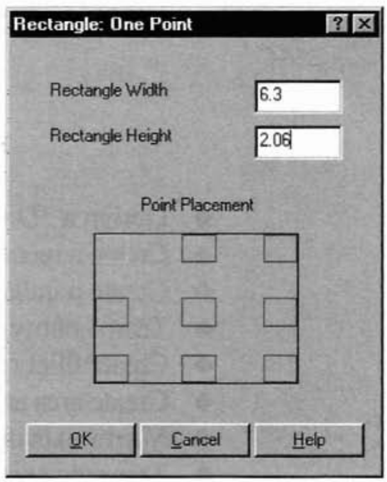
Objectives

- Design a 3D wireframe drawing.
- Create a rectangle using the 1 point entry method.
- Create parallel lines by defining the offset direction and distance.
- Trim 1 entity to another entity.
- Create fillet radii.
- Create arcs and lines using polar positioning.
- Mirror existing geometry to complete a part.
- Translate existing geometry to complete a part.
- Create a 2D milling toolpath.
- Perform solid model verification of the toolpath.



Geometry creation

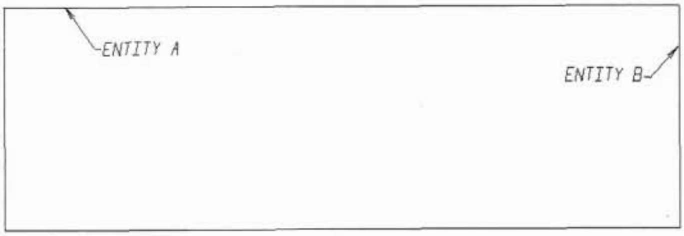
▶ *Create ¼ of the outside boundary*



1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Rectangle**
   - ◆ **1 point**
2. Enter the values shown in the dialog box at right, including width, height and point placement (upper left corner).
3. Choose **OK**.
4. Choose **Origin**.
5. Press [Esc] to exit the function.

▶ *Create the inside geometry*

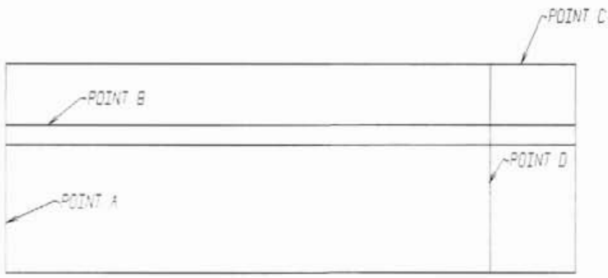
1. Choose the **Fit** button on the toolbar to center the geometry.
2. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Line**
   - ◆ **Parallel**
   - ◆ **Side/dist**
3. Select entity **A**.



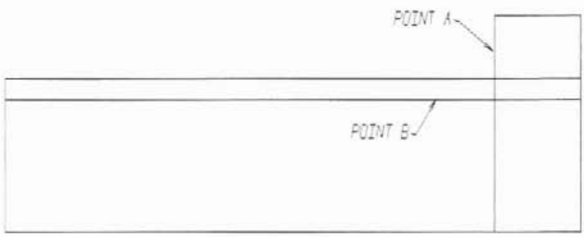
4. Indicate the offset direction by clicking anywhere inside the rectangle.
5. *Enter the parallel line distance.* **.6**
6. Select entity **A** again.
7. Click inside the rectangle below the offset line you just created.
8. *Enter the parallel line distance.* **.8**
9. Select entity **B**.
10. Click inside the rectangle.
11. *Enter the parallel line distance.* **.940**

▶ *Trim and fillet*

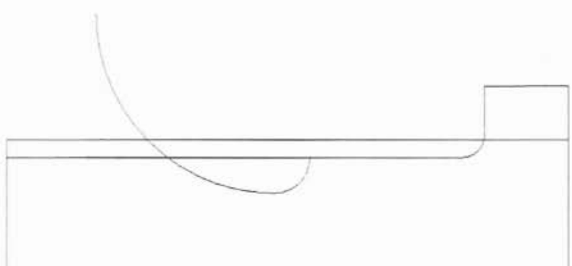
1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Modify**
   - ◆ **Trim**
   - ◆ **1 Entity**



2. Select point **A** as the entity to trim.
3. Select point **B** as the entity to trim to.
4. Select point **C** as the entity to trim.
5. Select entity **D** as the entity to trim to.

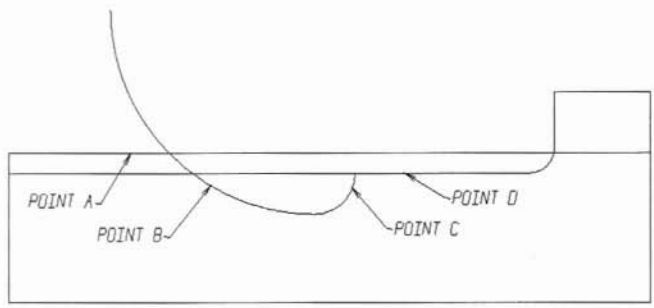


6. Choose
   - **BACKUP**
   - **BACKUP**
   - **Fillet**
   - **Radius**
7. *Enter the fillet radius.* **.250**
8. Select point **A**.
9. Select point **B**.



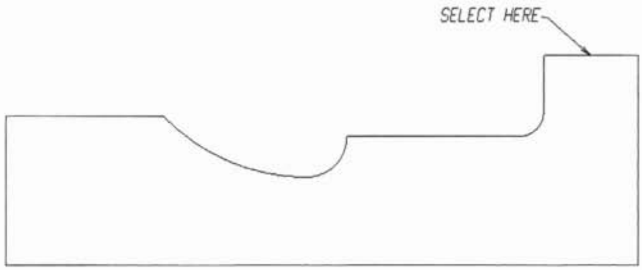
10. Choose
    - **MAIN MENU**
    - **Create**
    - **Arc**
    - **Polar**
    - **Center pt**
11. *Enter coordinates.* **3, 0.8**
12. *Enter the radius.* **2**

13. *Enter the initial angle.* **180**
14. *Enter the final angle.* **270**
15. *Enter the coordinates.* **3, -0.8**
16. *Enter the radius.* **0.4**
17. *Enter the initial angle.* **270**
18. *Enter the final angle.* **0**
19. Choose
    - **MAIN MENU**
    - **Modify**
    - **Trim**
    - **2 entities**



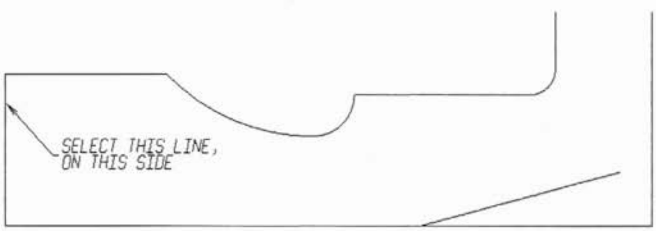
20. Select point **A** as the entity to trim.
21. Select point **B** as the entity to trim to.
22. Select point **C** the entity to trim.
23. Select point **D** the entity to trim to.
24. Choose
    - **MAIN MENU**
    - **Delete**

25. Select the horizontal line shown.

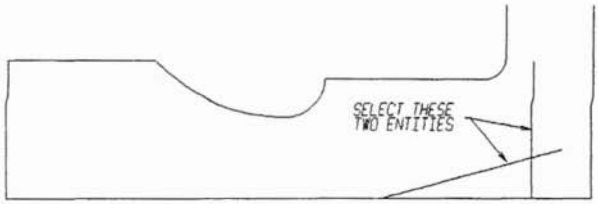


▶ *Create the outside profile*

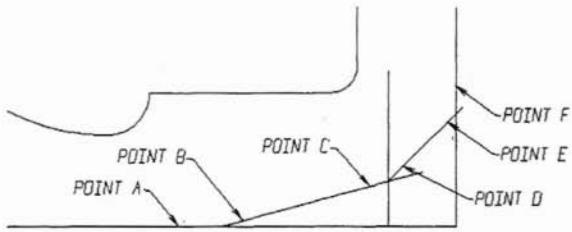
1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Line**
   - ◆ **Polar**
2. *Enter an endpoint.* **4.06, -2.06**
3. *Enter the angle in degrees.* **15**
4. *Enter the line length.* **2**
5. Choose
   - ◆ **BACKUP**
   - ◆ **Parallel**
   - ◆ **Side/dist**
6. Select the vertical line shown.



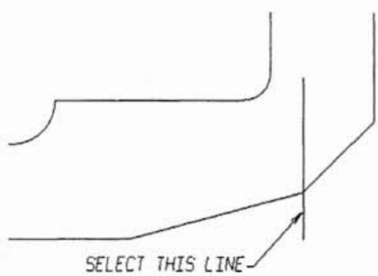
7. Indicate the offset direction by clicking to the right of the line.
8. *Enter the parallel line distance.* **5.66**
9. Choose
   - ◆ **BACKUP**
   - ◆ **BACKUP**
   - ◆ **Polar**
   - ◆ **Intersec**
10. Select the construction line and the 15 degree line as shown at right.



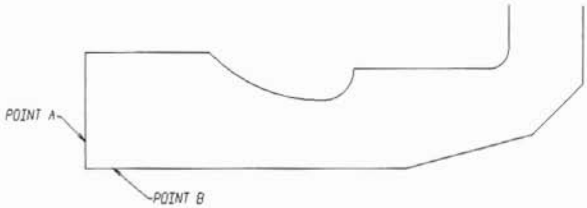
11. *Enter the angle in degrees.* **45**
12. *Enter the line length.* **1**
13. Choose
    - ◆ **MAIN MENU**
    - ◆ **Modify**
    - ◆ **Trim**
    - ◆ **2 entities**
14. Select point **A** as the entity to trim.
15. Select point **B** as the entity to trim to.



16. Select point **C** as the entity to trim.
17. Select point **D** as the entity to trim to.
18. Select point **E** as the entity to trim.
19. Select point **F** as the entity to trim to.
20. Choose
    - ◆ **MAIN MENU**
    - ◆ **Delete**
21. Select the vertical construction line.



22. Choose
    - **MAIN MENU**
    - **Create**
    - **Fillet**
    - **Radius**



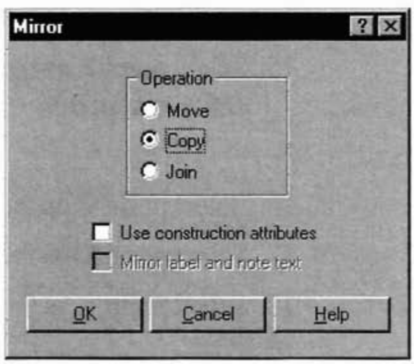
23. *Enter the fillet radius.* **.56**
24. Select point **A**.
25. Select point **B**.

▶ *Create the arcs representing the drilled holes*

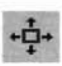
1. Choose
    - **MAIN MENU**
    - **Create**
    - **Arc**
    - **Circ pt+dia**
2. *Enter the diameter.* **.5**
3. *Enter the coordinates.* **.56, -1.5**
4. *Enter the coordinates.* **4.06, -1.5**

▶ *Transform the geometry to represent the whole part*

1. Choose
    - **MAIN MENU**
    - **Xform**
    - **Mirror**
    - **All**
    - **Entities**
    - **Done**
    - **X axis**



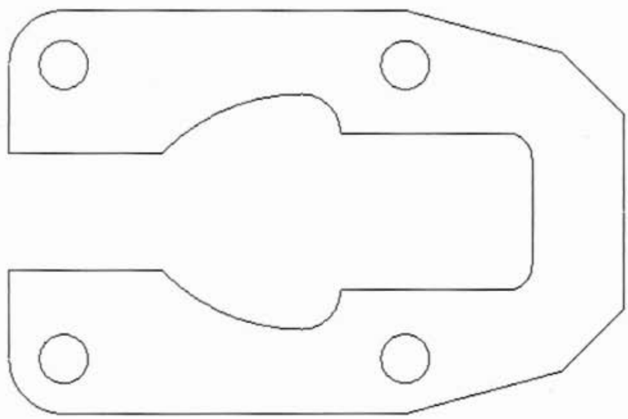
2. Enter the values shown in the dialog box at right.
3. Choose **OK**.



4. Choose the **Fit** button on the toolbar to fit the geometry.

5. Choose the **Clear colors** button on the toolbar.

The part should look like the following picture.

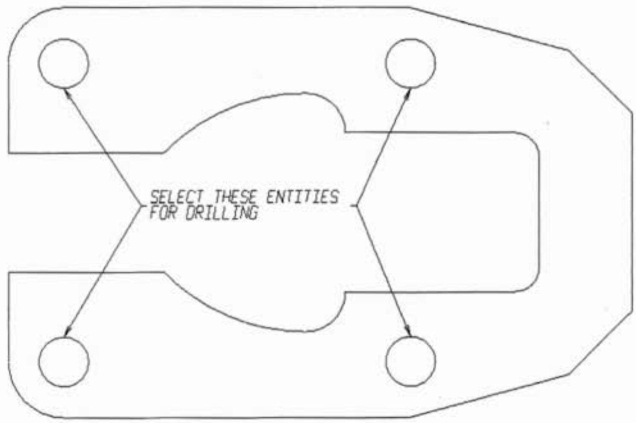


▶ *Save the file*

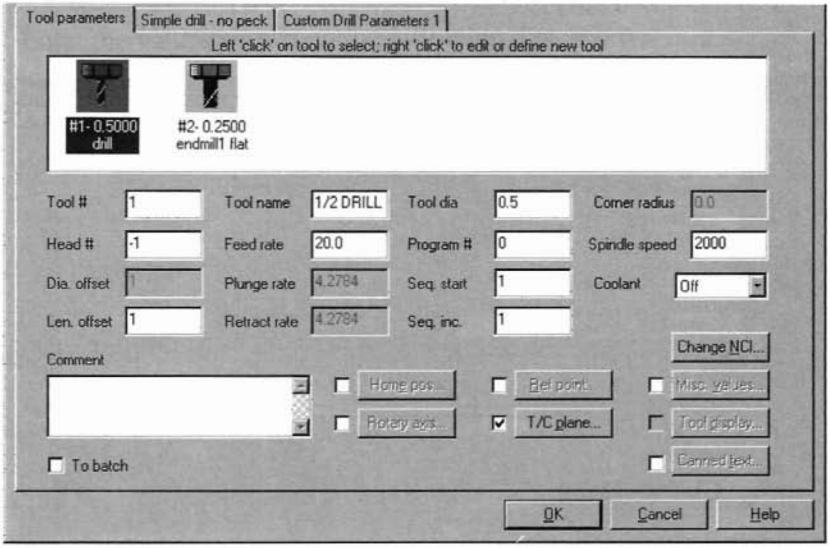
1. Choose
   - ◆ **MAIN MENU**
   - ◆ **File**
   - ◆ **Save**
2. Enter your name followed by the number 8.
3. Choose **Save**.

Toolpath creation

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Toolpaths**
   - ◆ **Drill**
   - ◆ **Manual**
   - ◆ **Center**
2. Select the top-left circle on the part in the following picture.

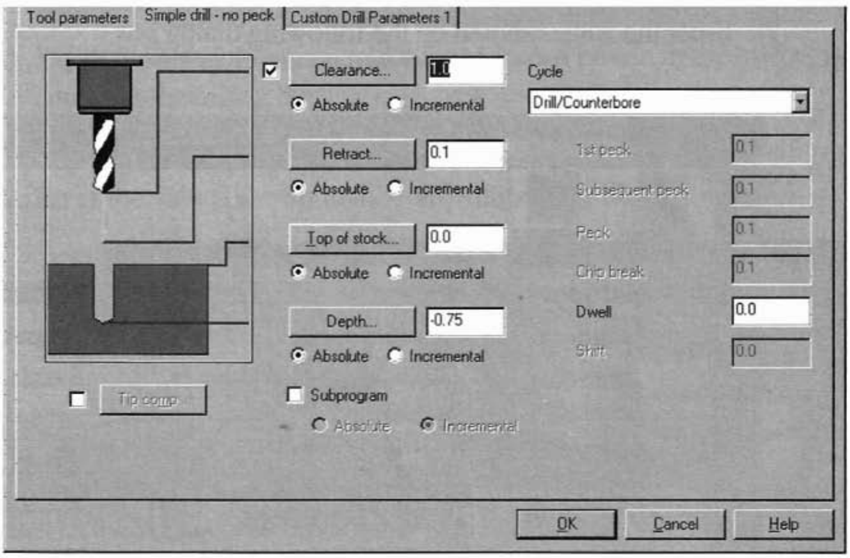


3. Choose **Center**.
4. Select another circle.
5. Repeat for the remaining two circles.
6. Press [Esc].
7. Choose **Done**.
8. Right-click in the tool display area.
9. Select **Get tool from library**.
10. Choose the 1/2" drill.
11. Choose **OK**.
12. Enter the values shown on the following dialog box.



13. Choose the **Simple drill –no peck** tab.

14. Enter the values shown on the following dialog box.

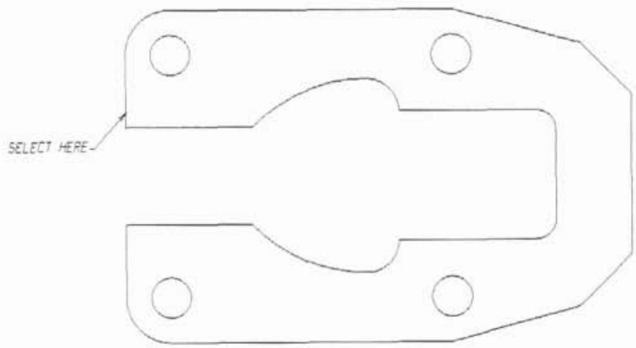


15. Choose **OK**.

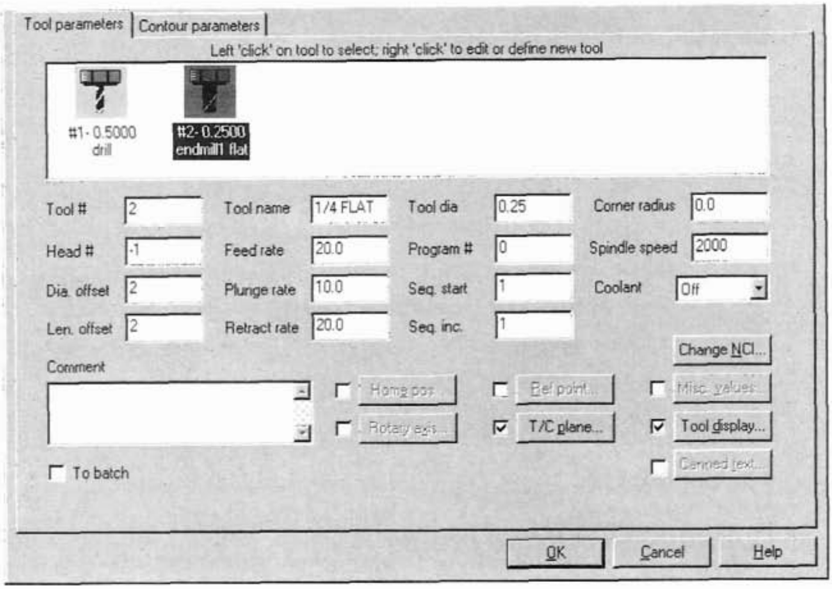
▶ *Contour the outside profile*

1. Choose **Contour**.

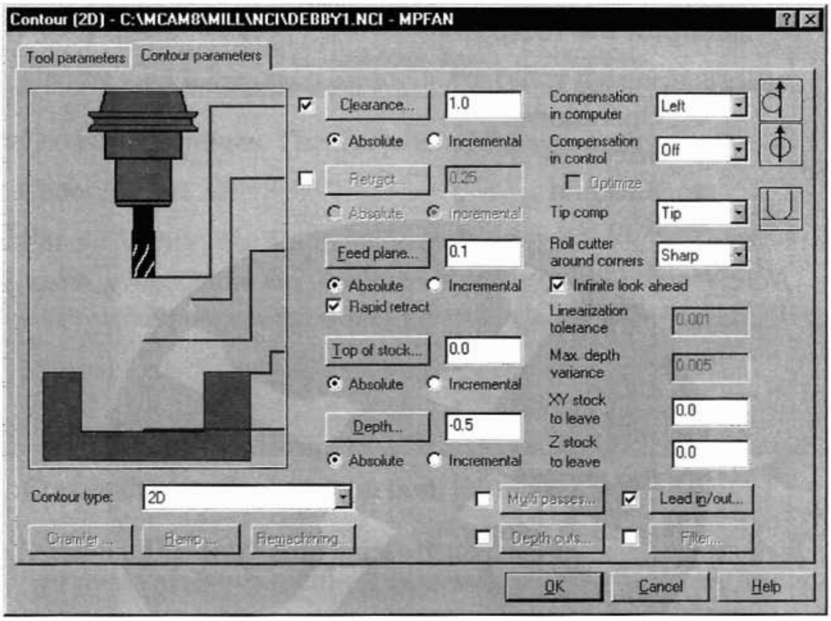
2. Select a point as shown in the following picture to begin chaining.



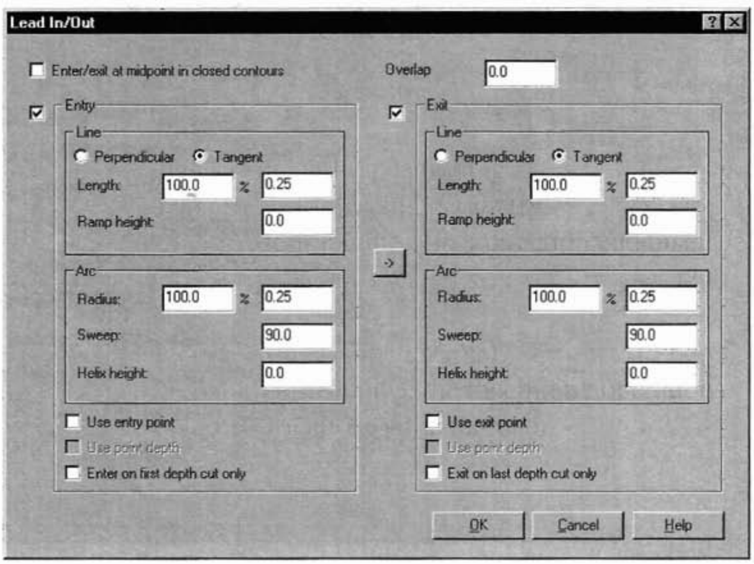
3. Choose **Done**.
4. Right-click in the tool display area.
5. Select **Get tool from library**.
6. Select the ¼" flat endmill and choose **OK**.
7. Enter the values shown on the following dialog box.



8. Select the **Contour parameters** tab.
9. Enter the values shown on the following dialog box.



10. Select the check box next to the **Lead in/out** button in the lower-right corner of the dialog box.
11. Choose the **Lead in/out** button.
12. Enter the values shown in the following dialog box.



13. Choose **OK**.
14. Choose **OK**.

▶ *Backplot the toolpath*

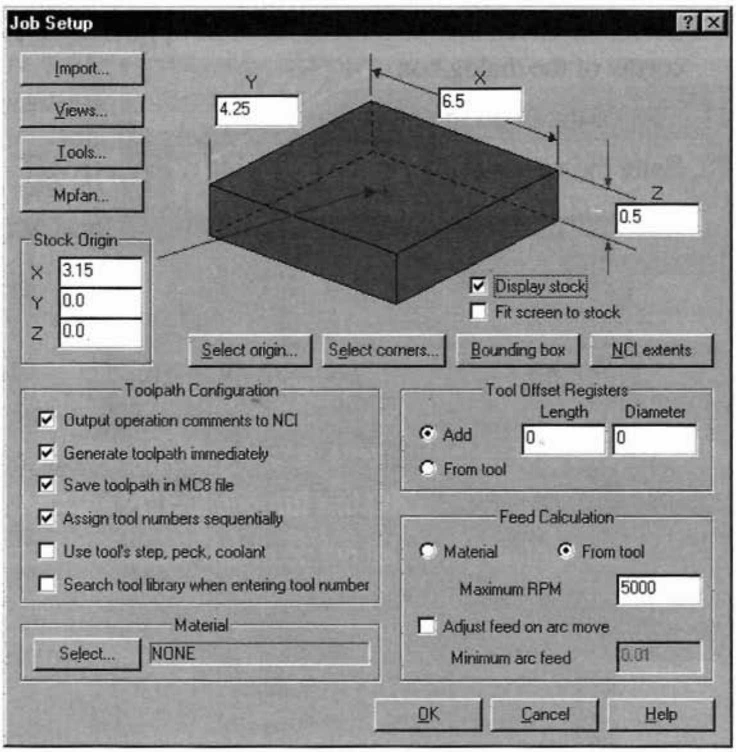
1. Choose
   - **Operations**
   - **Select All**
   - **Backplot**
   - **Run**

Note: If you'd like to watch the toolpath run more slowly, select Step instead of Run. Hold the ***S*** *key down to move through the toolpath.*

2. Choose **BACKUP** to return to the Operations Manager.
3. Choose **OK** to exit the Operations Manager.

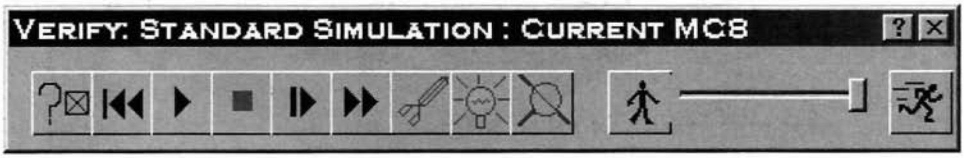
▶ *Verify the program*

1. Choose **Job setup**.
2. Enter the values shown on the following dialog box.

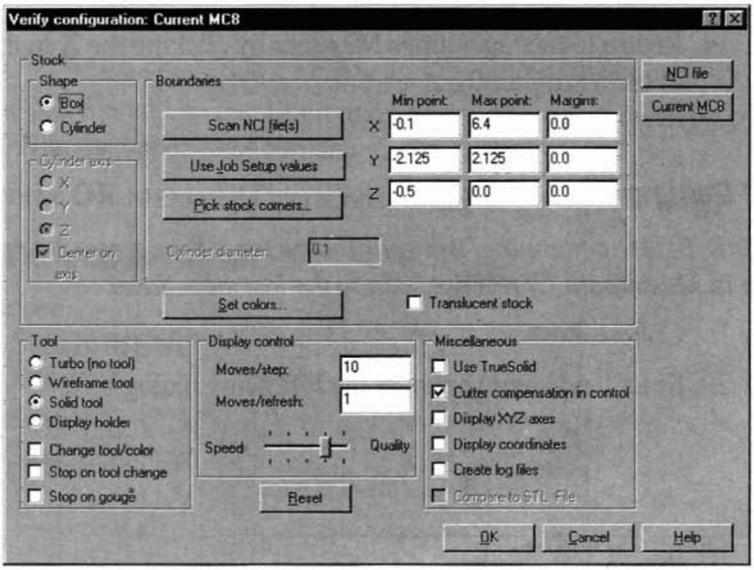


3. Choose **OK**.
4. Choose **Gview** from the Secondary Menu.
5. Choose **Isometric**.

6. Choose the **Fit** button on the toolbar to center the geometry.
7. Choose **Operations**. The Operations Manager opens.
8. Choose **Select All**.
9. Choose **Verify**. The Verification toolbar opens.



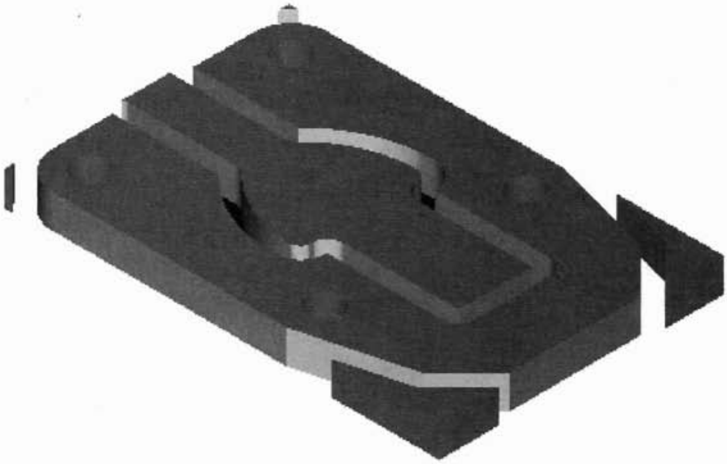
10. Choose the **Configure** button from the toolbar.
11. Select the **Use Job Setup** values button and enter the values shown.



12. Choose **OK** to exit the dialog box.

13. Choose the **Machine** button from the Verify toolbar.

The verified part should look like the following picture.

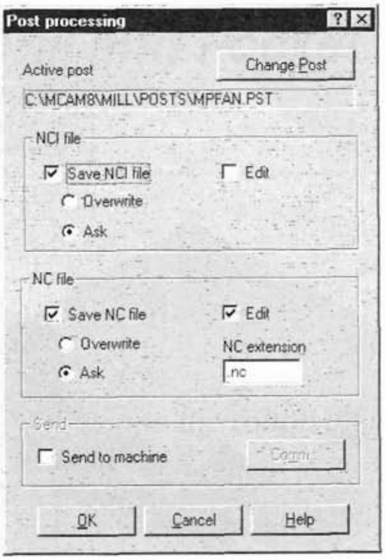


14. Return to the Operations Manager by clicking the **X** in the right corner of the Verify toolbar.

▶ *Posting the code*

Note: Before continuing, ask your instructor if the correct post processor is set up in Mastercam. If it is not, skip to the last step, Save.

1. Choose **Post**.
2. Enter the values shown in the following dialog box.

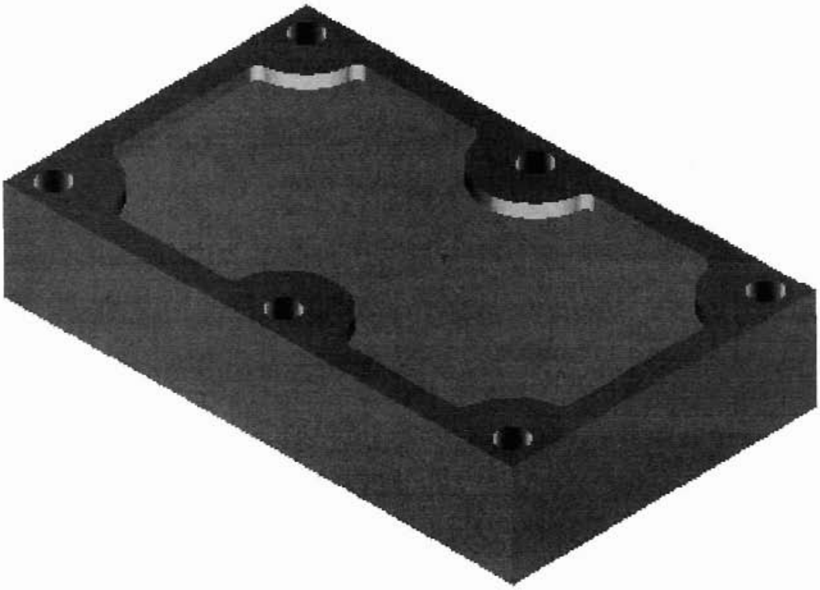


3. Choose
   - **OK**
   - **Save**
   - **Save**
4. The NC file will be displayed in an editor. Close the screen by selecting the **X** in the upper right corner.
5. Choose **OK** to exit the Operations Manager.

▶ *Save the MC8 file again*

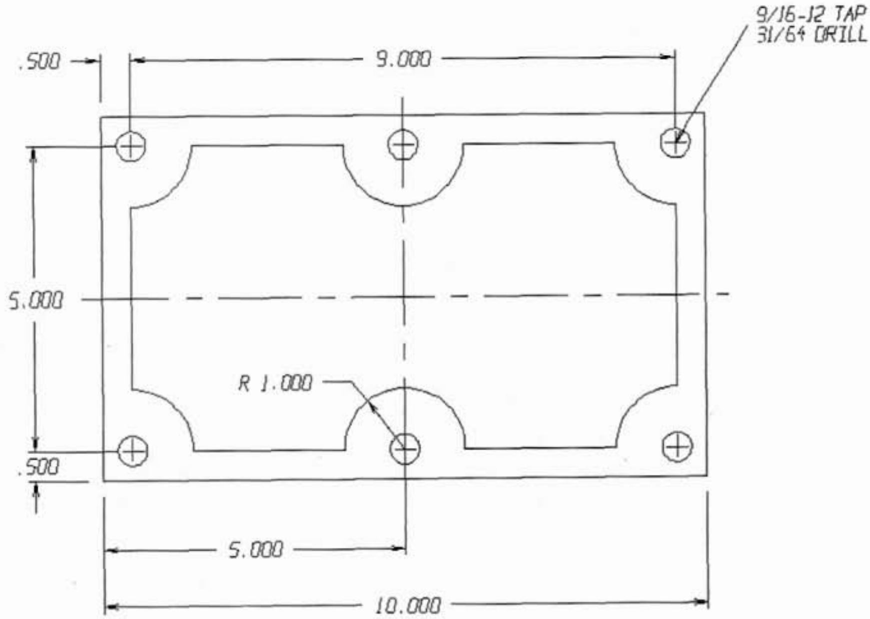
1. Choose
   - **MAIN MENU**
   - **File**
   - **Save**
   - **Save**
   - **Yes**

9 2D Drill and Pocket



Objectives

- Design a 3D wireframe drawing.
- Create a rectangle using the 2 point entry method.
- Create parallel lines by defining the offset direction and distance.
- Create lines using coordinate entry functions.
- Create arcs using center/radius functions.
- Create arcs using polar positioning.
- Trim two entities together.
- Mirror existing geometry to complete a part.
- Translate existing geometry to create a 3-dimensional block.
- Create 2D milling toolpaths.
- Perform solid model verification of the toolpath.



Geometry creation

▶ *Create ¼ of the outside boundary*

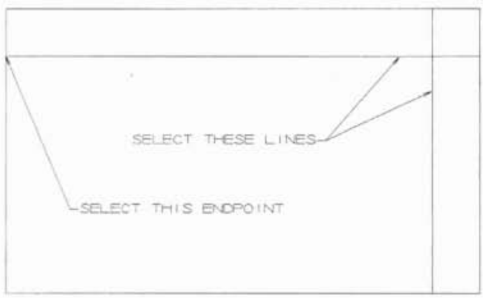
1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Rectangle**
   - ◆ **2 points**
2. *Enter the coordinates.* **0, 0**
3. *Enter the coordinates.* **5, 3**

4. Choose the **Fit** button on the toolbar to center the geometry.

▶ *Create the inside entities*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Line**
   - ◆ **Parallel**
   - ◆ **Side/dist**
2. Select the vertical line on the right.
3. Indicate the offset direction by clicking anywhere inside the rectangle.
4. *Enter the parallel line distance.* **.5**

5. Select the top horizontal line.
6. Indicate the offset direction by clicking anywhere inside the rectangle.
7. *Enter the parallel line distance.* **.5**
8. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Arc**
   - ◆ **Polar**
   - ◆ **Center pt**
   - ◆ **Intersec**
9. Select the two intersecting lines.

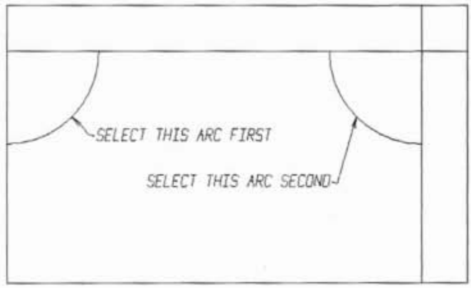


10. *Enter the radius.* **1**
11. *Enter the initial angle.* **180**
12. *Enter the final angle.* **270**
13. Select the endpoint of the line shown.

14. *Enter the radius.* **1**
15. *Enter the initial angle.* **270**
16. *Enter the final angle.* **0**

▶ *Create the arcs to represent the drill holes*

1. Choose **Center**.
2. Select the left arc.
3. *Enter the radius.* **31/64/2**

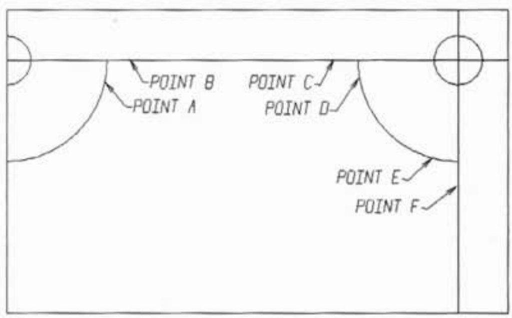


Note: Mastercam does the calculation, dividing 31 by 64 by 2.

4. *Enter the initial angle.* **270**
5. *Enter the final angle.* **90**
6. Choose
   - ◆ **BACKUP**
   - ◆ **BACKUP**
   - ◆ **Circ pt+dia**
7. *Enter the diameter.* **31/64**
8. Choose **Center**.
9. Select the right arc.

▶ *Trim the entities*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Modify**
   - ◆ **Trim**
   - ◆ **2 entities**
2. Select point **A** as the entity to trim to.
3. Select point **B** as the entity to trim.

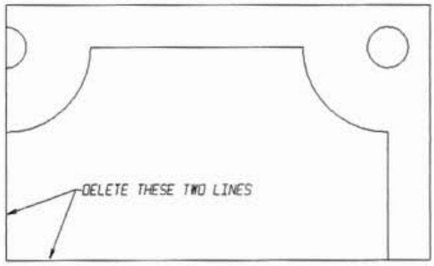


4. Select point **C** as the entity to trim to.

5. Select point **D** as the entity to trim.
6. Select point **E** as the entity to trim.
7. Select point **F** as the entity to trim to.

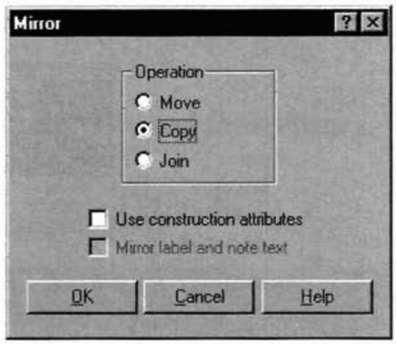
▶ *Delete the construction lines*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Delete**
2. Select the horizontal line located at **YO**.
3. Select the vertical line located at **XO**.

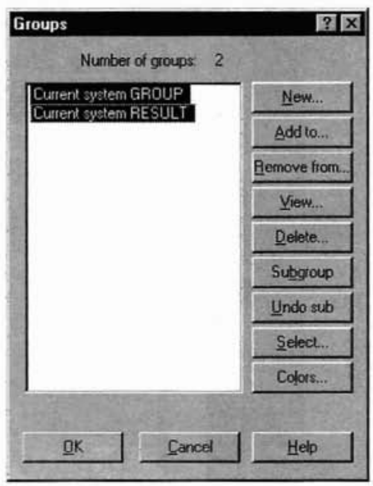


▶ *Mirror the current geometry*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Xform**
   - ◆ **Mirror**
   - ◆ **All**
   - ◆ **Entities**
   - ◆ **Done**
   - ◆ **X axis**



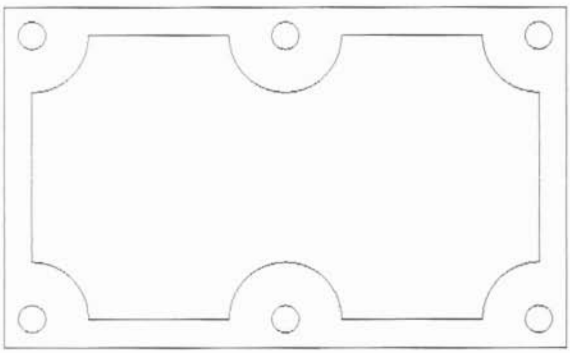
2. Choose **Copy** in the Mirror dialog box.
3. Choose **OK**.
4. Choose **Group**.
5. In the dialog box that opens, select **Current system GROUP**. Press [Ctrl] and select **Current system Result** as shown at right.
6. Choose **OK**.
7. Choose
   - ◆ **Done**
   - ◆ **Y Axis**



8. Choose **Copy** in the Mirror dialog box.
9. Choose **OK**.
10. Choose
    - **Done**
    - **MAIN MENU**
    - **Screen**
    - **Clr colors**

11. Choose the **Fit** button on the toolbar to center the geometry.

The part should look like the following picture.



12. Choose
    - **MAIN MENU**
    - **File**
    - **Save**
13. Enter your name, followed by 9.
14. Choose **Save**.

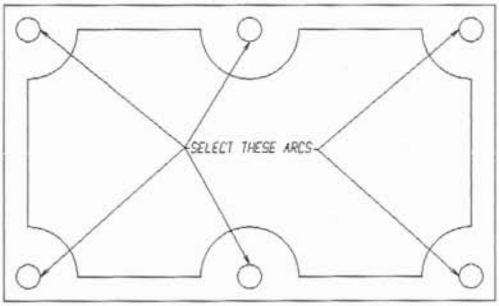
Toolpath creation

Drill 6 holes

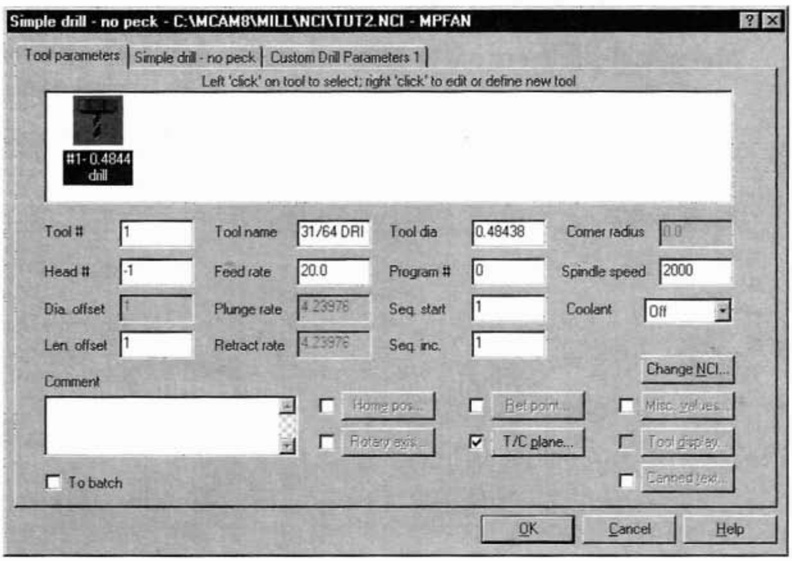
1. Choose
    - **MAIN MENU**
    - **Toolpaths**
    - **Drill**

- **Manual**
- **Center**

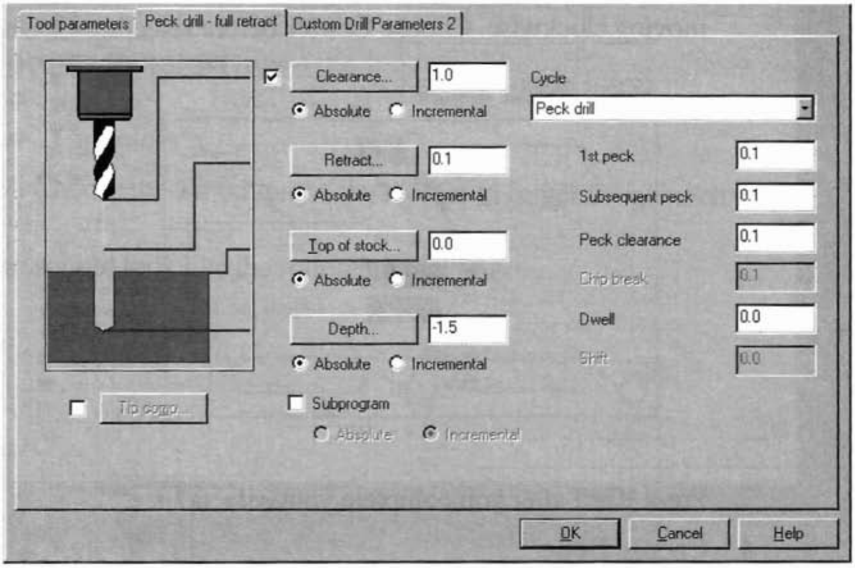
2. Select the arcs to drill beginning with the one in the top left corner and moving clockwise. Choose **Center** before selecting each arc to drill.



3. Press [Esc] after you complete your selections.
4. Choose **Done**.
5. Right-click in the tool display area and select **Get tool from library**.
6. Select the 31/64 " drill and choose **OK**.
7. Enter the values shown on the following dialog box.



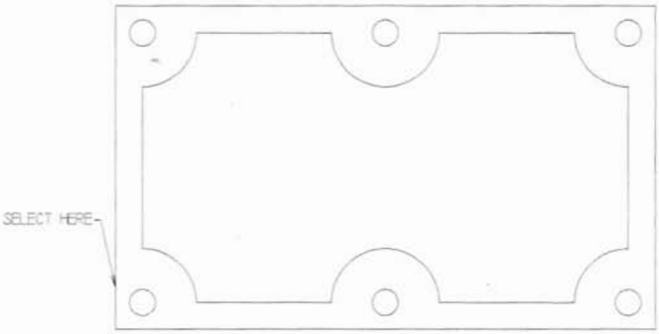
8. Select the **Simple drill/no peck** tab at the top of the dialog box.
9. Enter the values shown on the following dialog box.



10. Choose **OK**.

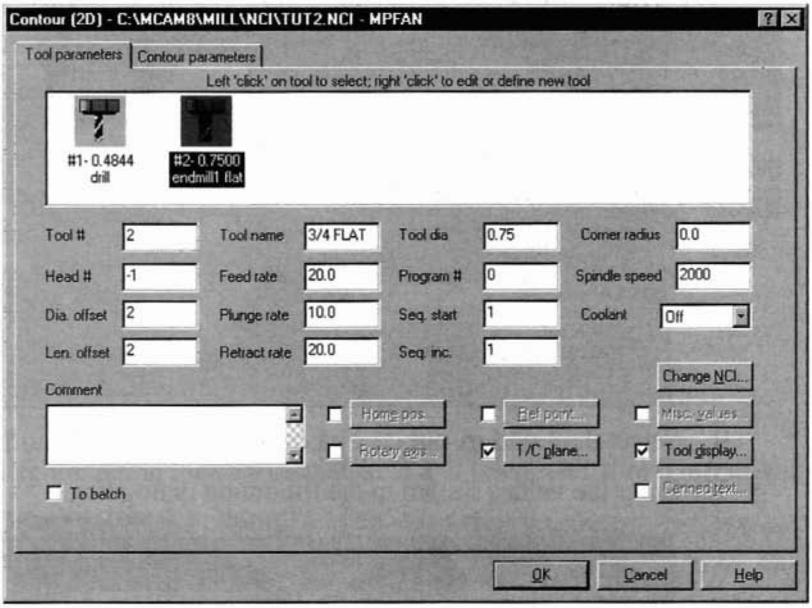
▶ *Contour the outside profile*

1. Choose
   - ◆ **Contour**
   - ◆ **Chain**
2. Select an entity in the outside profile.



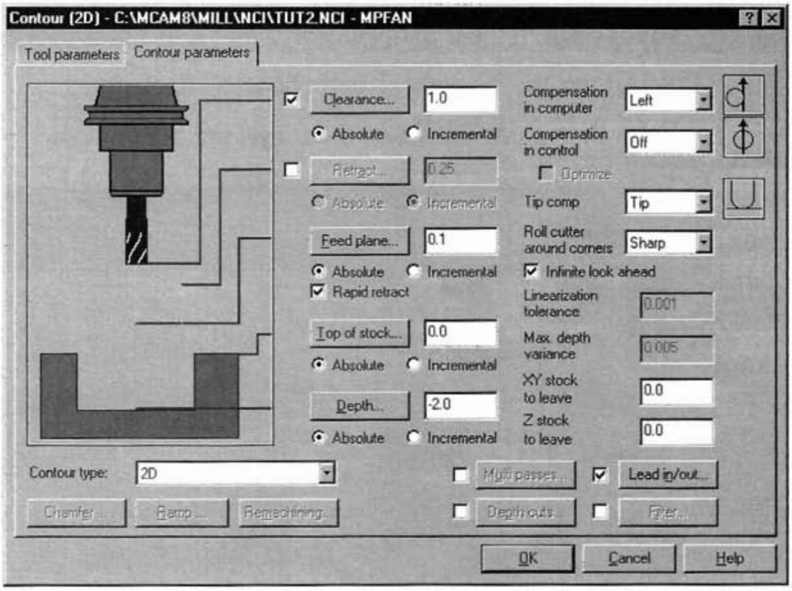
Note: The chain should be moving in a clockwise direction.

3. Choose **Done**.
4. Right-click in the tool display area and select **Get tool from library**.
5. Select the ¾" flat endmill and choose **OK**.
6. Enter the values shown on the following dialog box.



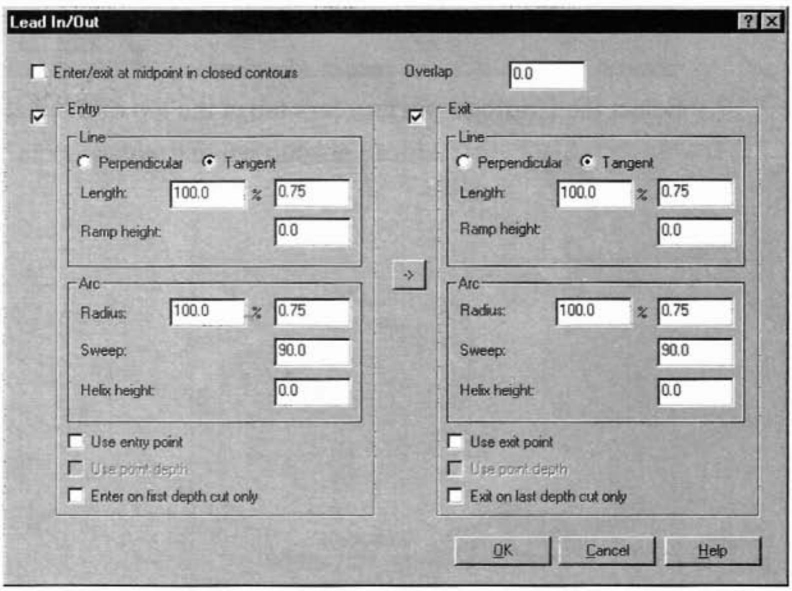
7. Select the **Contour parameters** tab at the top of the dialog box.

8. Enter the values shown in the following dialog box.



9. Select the **Lead in/out** check box and choose the **Lead in/out** button.

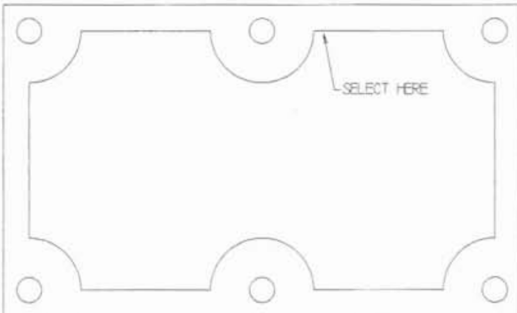
10. Enter the values shown in the following dialog box.



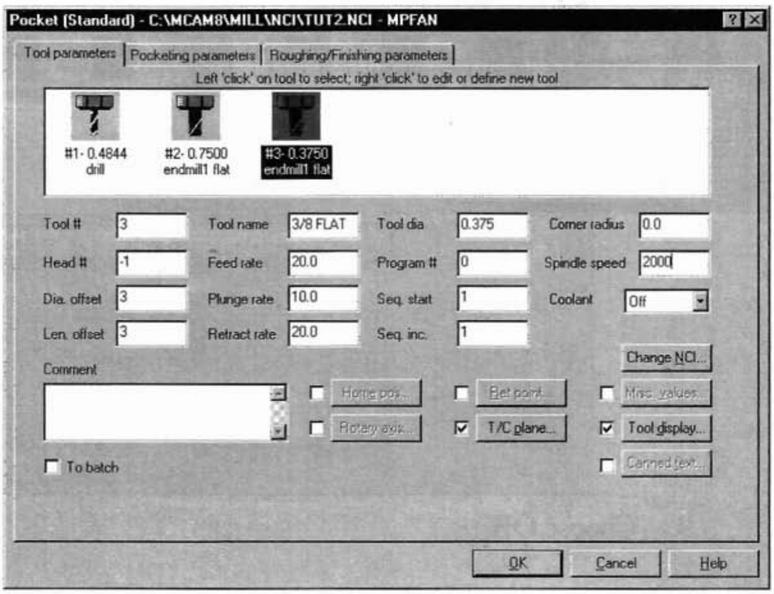
11. Choose **OK** twice.

▶ *Pocket the inside geometry*

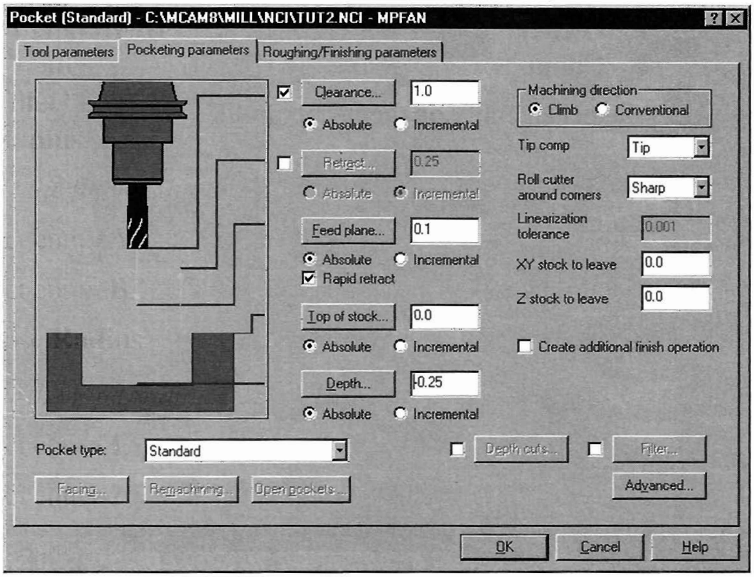
1. Choose
   - ◆ **Pocket**
   - ◆ **Chain**
2. Select an entity on the inside profile.



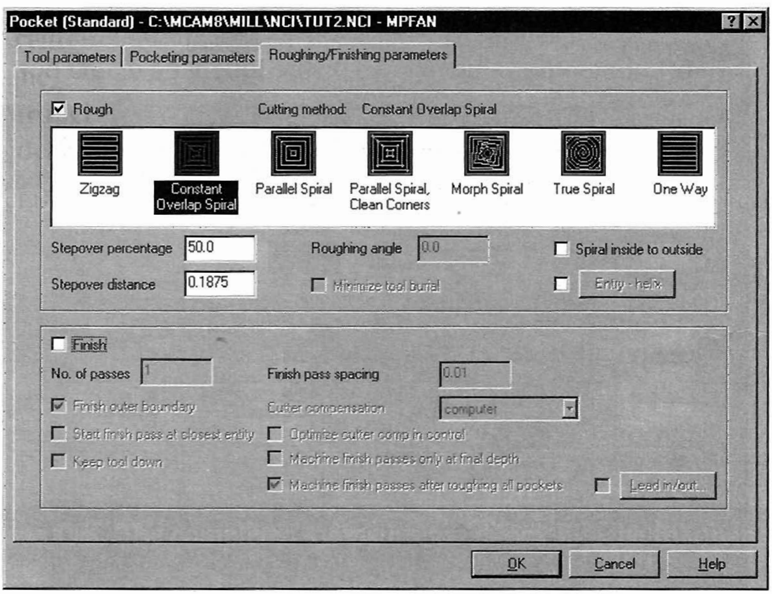
3. Choose **Done**.
4. Right-click in the tool display area and select **Get tool from library**.
5. Select a 3/8" flat endmill and choose **OK**.
6. Enter the values shown on the following dialog box.



7. Select the **Pocketing parameters** tab and enter the values shown on the following dialog box.

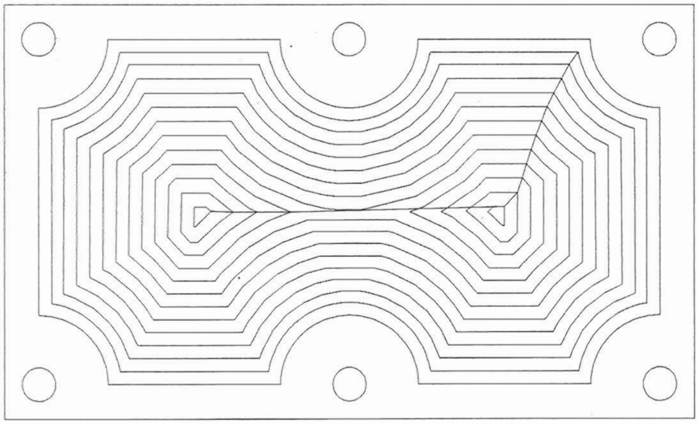


8. Select the **Roughing/Finishing parameters** tab and enter the values shown on the following dialog box.



9. Choose **OK**.

The pocket toolpath should look like the following picture.



▶ *Backplot the toolpath*

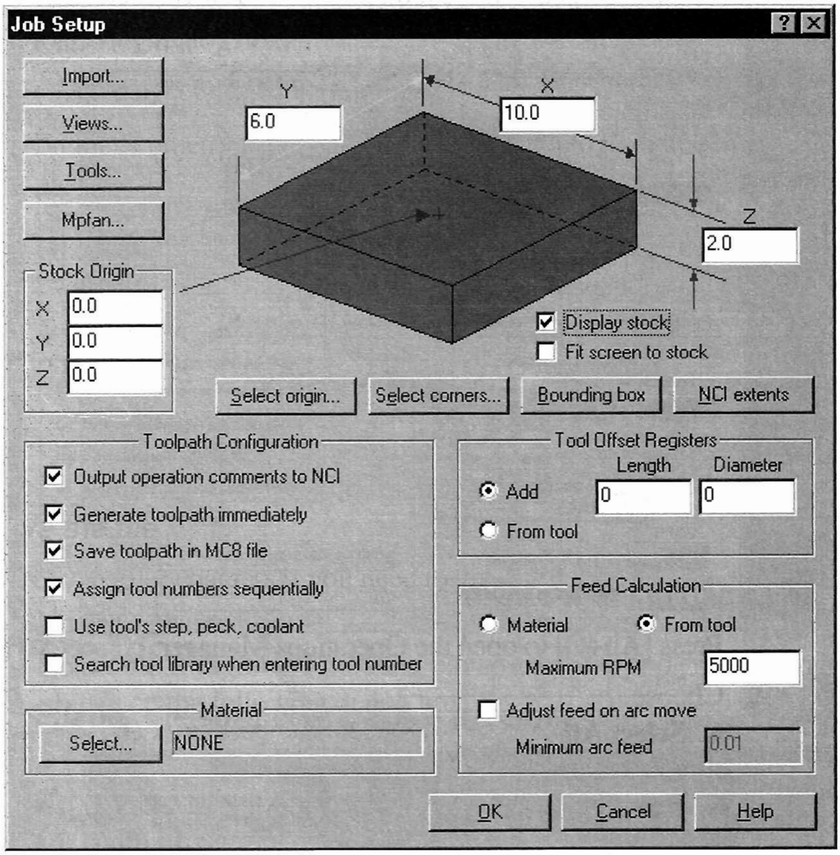
1. Press [Alt+O] to open the Operations Manager.
2. Choose
   - ◆ **Select All**
   - ◆ **Backplot**
   - ◆ **Run**

Note: Select ***Step*** *instead of* ***Run*** *if you would like to watch the progress of the toolpath. Hold down the [S] key or keep choosing the* ***Step*** *menu option.*

3. Choose
   - ◆ **BACKUP**
   - ◆ **OK**

▶ *Set up the stock to be machined*

1. Choose **Job setup**.
2. Enter the values shown on the following dialog box.



3. Choose **OK**.

4. Choose the **Gview (isometric)** button from the toolbar.

5. Choose the **Fit** button from the toolbar to fit the geometry in the graphics window.

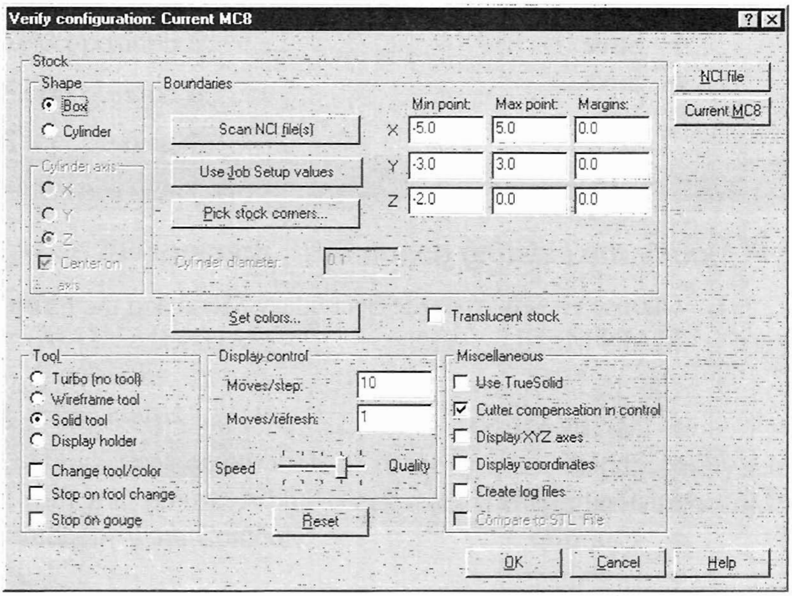
At this point you should have a 3D view of the part.

▶ *Verify*

1. Choose **Operations**.
2. Choose **Verify**.

3. Choose the **Configure** button from the toolbar.

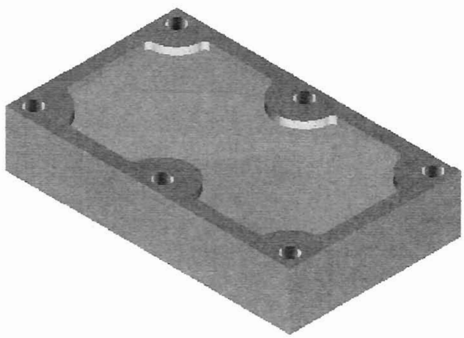
4. Select the **Use Job Setup values** button and enter the values shown on the following dialog box.



5. Choose **OK**.

6. Choose the **Machine** button from the Verify toolbar.

The part should look like the following picture.



7. Close the Verify toolbar by selecting the **X** in the right corner.

Save the updated MC8 file

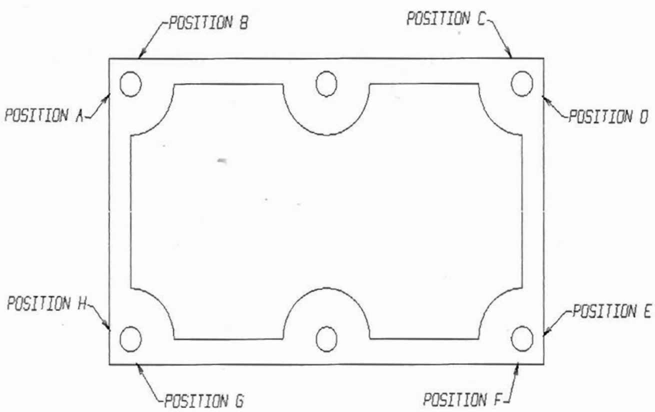
1. Choose **OK** to exit the Operations Manager.

2. Choose
   - ◆ **MAIN MENU**
   - ◆ **File**
   - ◆ **Save**
   - ◆ **Save**
   - ◆ **Yes**

Using associativity

▶ *Modify the existing geometry*

1. Choose **Gview** from the Secondary Menu and then **Top** from the MAIN MENU.
2. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Next menu**
   - ◆ **Chamfer**
   - ◆ **Distances**
3. *Enter the first distance.* **0.25**
4. *Enter the second distance.* **0.25**
5. Select position **A**.
6. Select position **B**.



7. Select position **C**.
8. Select position **D**.
9. Select position **E**.
10. Select position **F**.
11. Select position **G**.
12. Select position **H**.

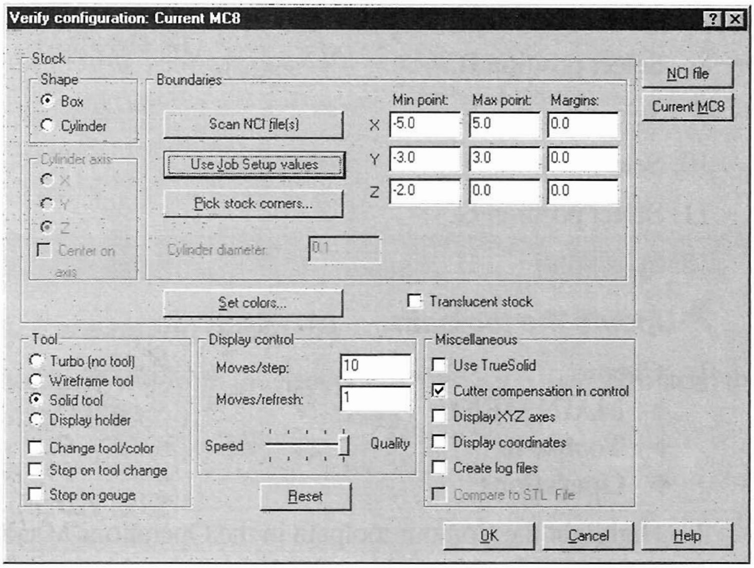
▶ *Update the toolpath*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Toolpaths**
   - ◆ **Operations**
2. Highlight the Contour toolpath in the Operations Manager by selecting it with your mouse. A blue check mark should appear in the folder beside the word Contour.
3. Choose
   - ◆ **Regen Path**
   - ◆ **Select All**

▶ *Verify the changed part*

1. Choose **Verify** from the Operations Manager.

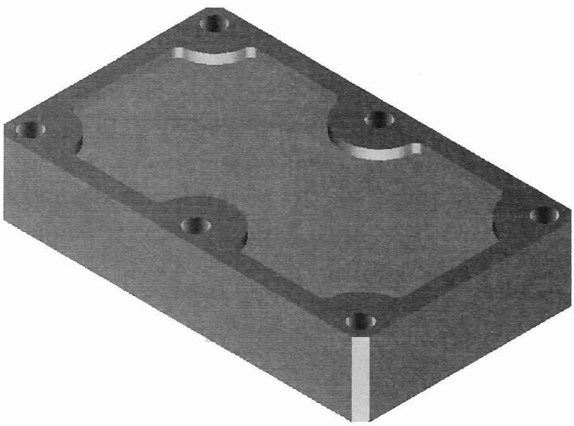
2. Choose the **Configure** button from the toolbar.
3. Select the **Use Job Setup Values** button and enter the values on the following dialog box.



4. Choose **OK**.

5. Choose the **Machine** button from the Verify toolbar.

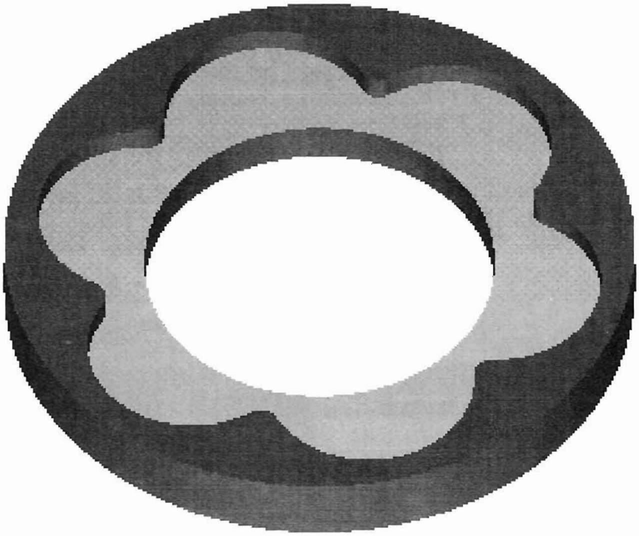
The finished part should look like the following picture.



▶ *Save the part*

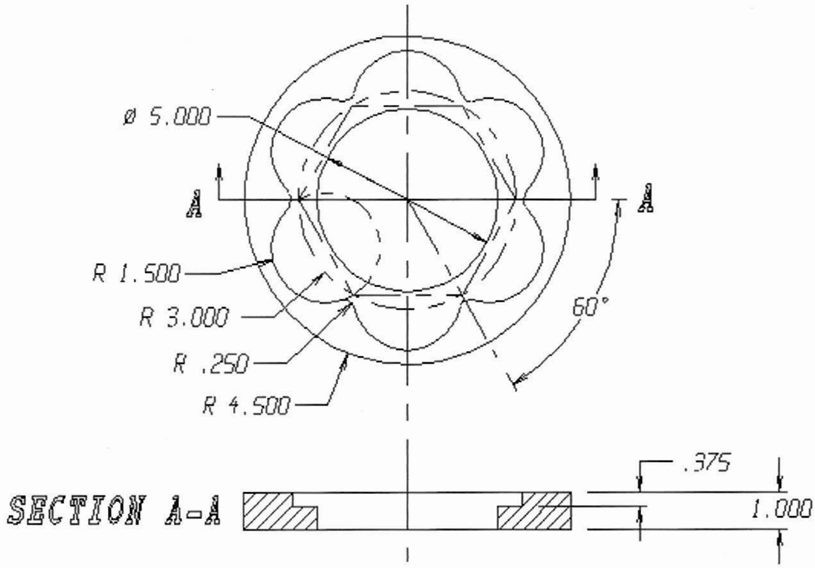
Refer to previous chapters if necessary.

10 2D Pocket Multiple Depths



Objectives

- Design a 3D wireframe drawing.
- Create an arc using coordinate positioning.
- Create a polygon.
- Create fillet radii.
- Create 2D pockets and contours.
- Use lead in and lead out toolpath entry.
- Define a 3D cylindrical block the size of the workpiece.
- Perform solid model verification of the toolpath.

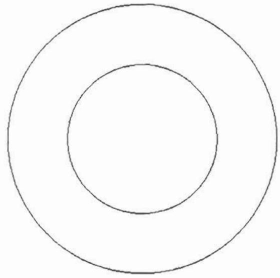


Geometry creation

▶ *Create the outside profile and the inner bore*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Arc**
   - ◆ **Circ pt+dia**
2. *Enter the diameter.* **9**
3. *Enter the coordinates.* **0,0**
4. Choose
   - ◆ **BACKUP**
   - ◆ **Circ pt dia**
5. *Enter the diameter.* **5**
6. *Enter the coordinates.* **0,0**

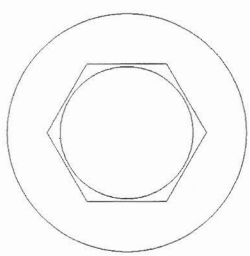
7. Choose the **Fit** button on the toolbar to center the geometry.
8. Your drawing should look like the following picture.



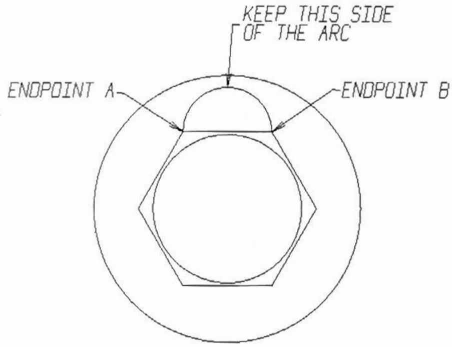
▶ *Create the polygon*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Next menu**
   - ◆ **Polygon**
   - ◆ **No. sides**
2. *Enter the number of sides.* **6**
3. Choose **Radius.**
4. Enter the radius. **3**
5. Choose **Start Angle.**
6. Enter the Start Angle. **0**
7. Toggle **Meas crnr** to **Y.**
8. Toggle **NURBS** to **N.**
9. Choose
   - ◆ **Do it**
   - ◆ **Center**
10. Select the center arc.

Your drawing should look like the following picture.



Create six 1.5 radius arcs



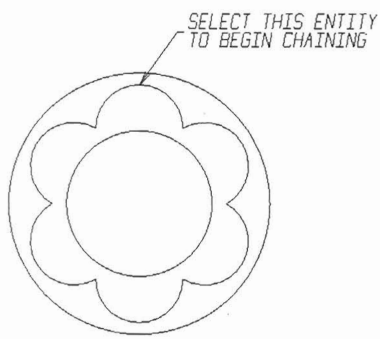
1. Choose
   - **MAIN MENU**
   - **Create**
   - **Arc**
   - **Endpoints**
   - **Endpoint**
2. Select Endpoint **A.**
3. Select Endpoint **B.**
4. *Enter the radius.* **1.5**
5. Select the portion of the arc to keep.
6. Create the remaining arcs by substituting endpoints and repeating steps 4 and 5.

Delete the construction lines

1. Choose
   - **MAIN MENU**
   - **Delete**
   - **All**
   - **Lines**

Create the corner fillets

1. Choose
   - **MAIN MENU**
   - **Modify**
   - **Fillet**
   - **Radius**
2. *Enter the radius.* **.25**
3. Choose **Chain.**
4. Select one of the 1.5 radius arcs.
5. Choose **Done.**



Save the file

1. Choose
   - **MAIN MENU**

- **File**
- **Save**

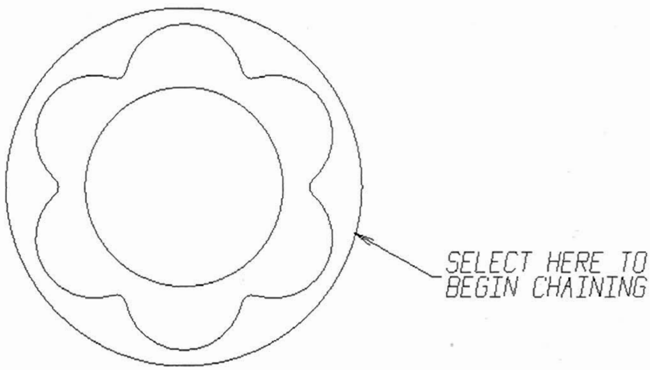
2. Save this file using your name and the number 10 (johnson10)

Toolpath creation

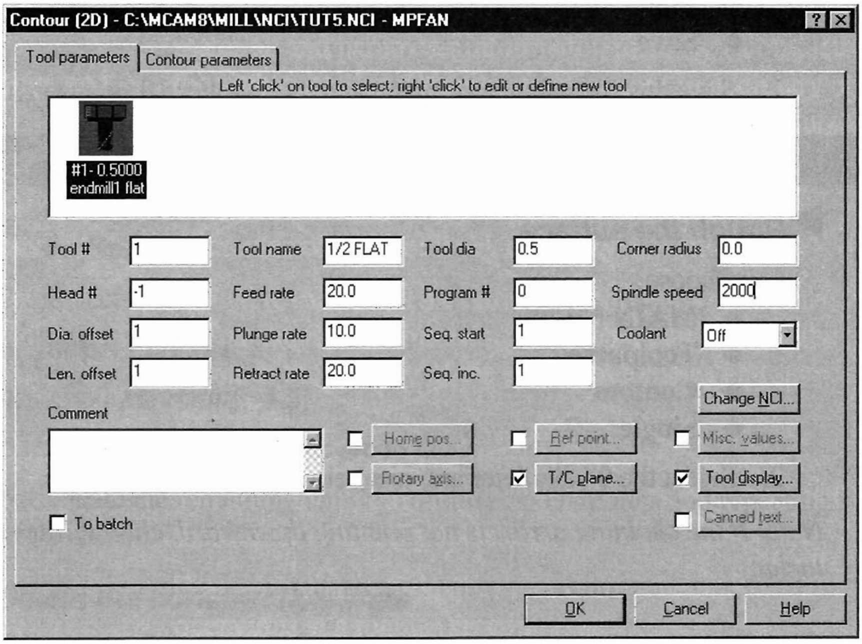
Rough the surface

1. Choose
   - **MAIN MENU**
   - **Toolpaths**
   - **Contour**
   - **Single**
2. Select the 9" diameter arc, just below 0 degrees.

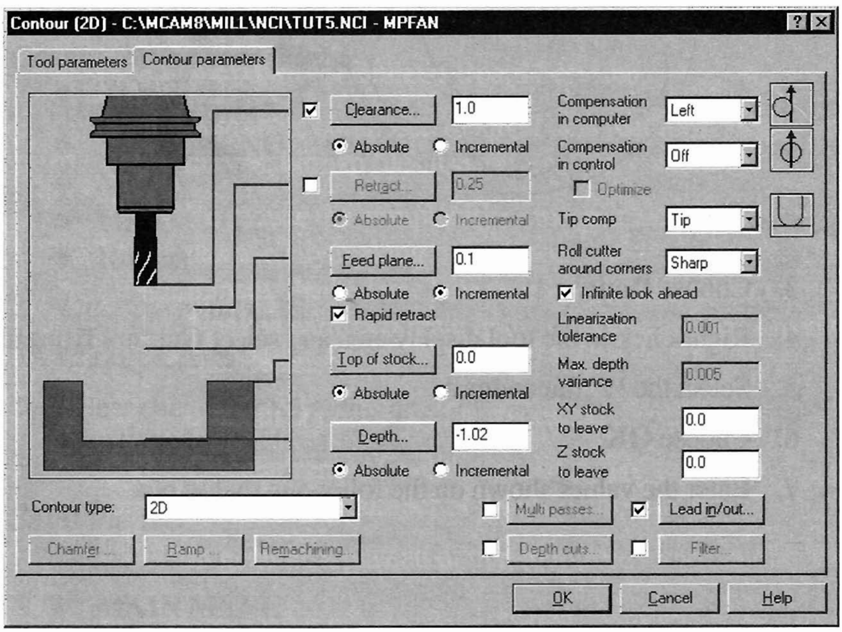
Note: If the chaining arrow is not pointing downward, choose Reverse from the menu.



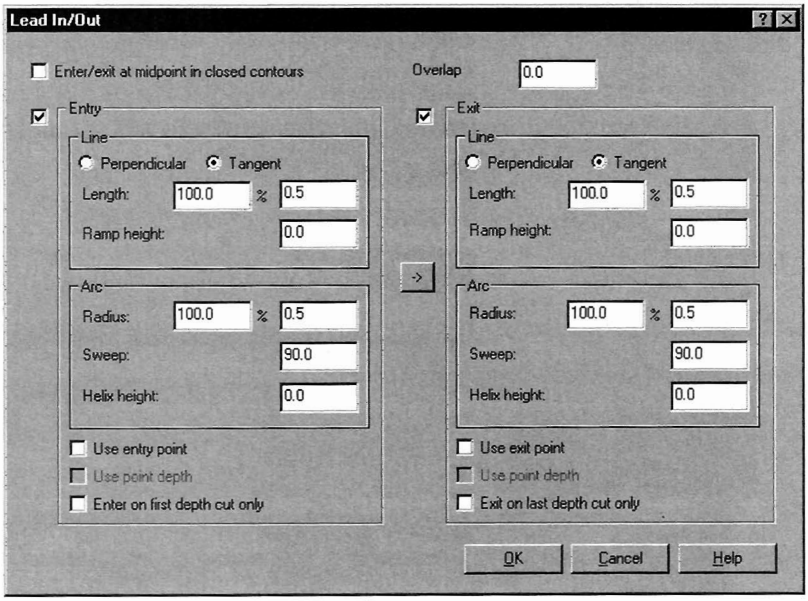
3. Choose **Done**.
4. Right-click in the tool display area and select **Get tool from library**.
5. Select the ½" flat endmill.
6. Choose **OK**.
7. Enter the values shown on the following dialog box.



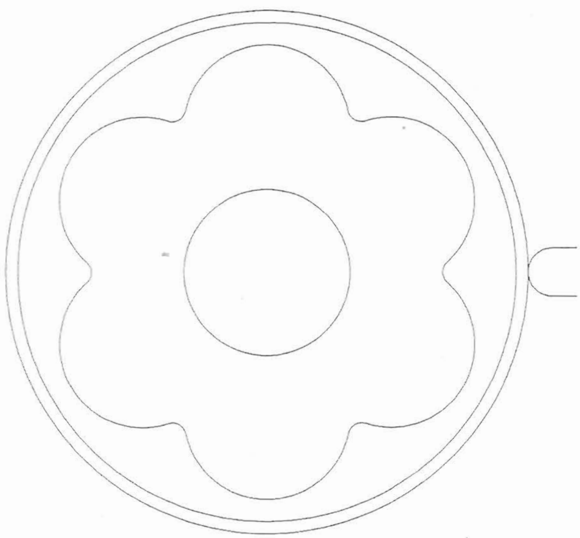
8. Select the **Contour parameters**
9. Enter the values shown on the following dialog box. Choose the **Lead in/out** button.



11. Enter the values shown on the following dialog box.
12. Choose **OK** twice.

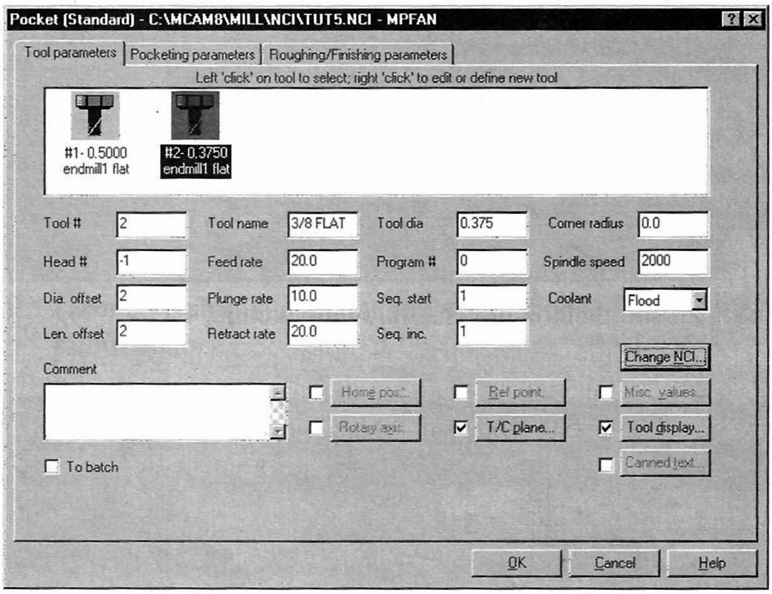


Your part should look like the following picture.

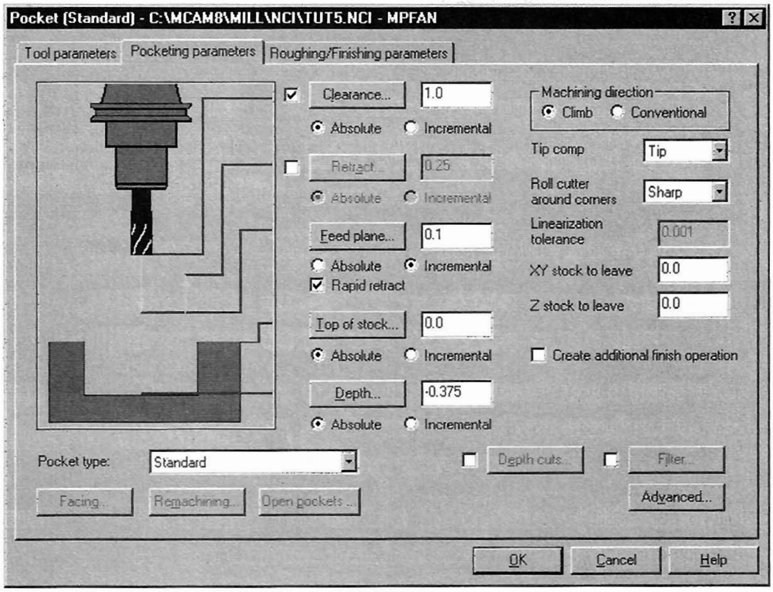


▶ *Pocket the 1.5 radius*

1. Choose **Pocket**.
2. Select any one of the 1.5" arcs.
3. Choose **Done**.
4. Right-click in the tool display area and select **Get tool from library**.
5. Select the 3/8" flat endmill.
6. Choose **OK**.
7. Enter the values shown on the following dialog box.

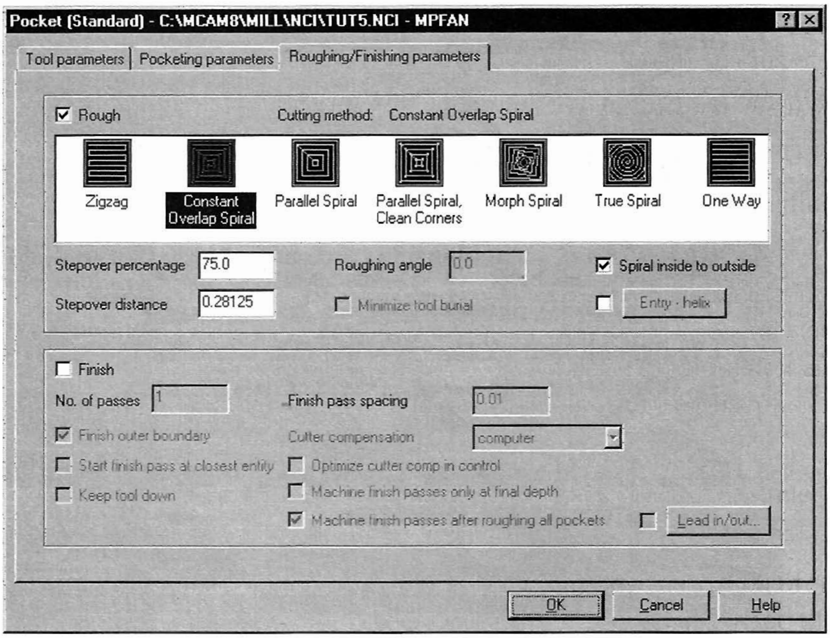


8. Select the **Pocketing parameters** tab.
9. Enter the values shown on the following dialog box.



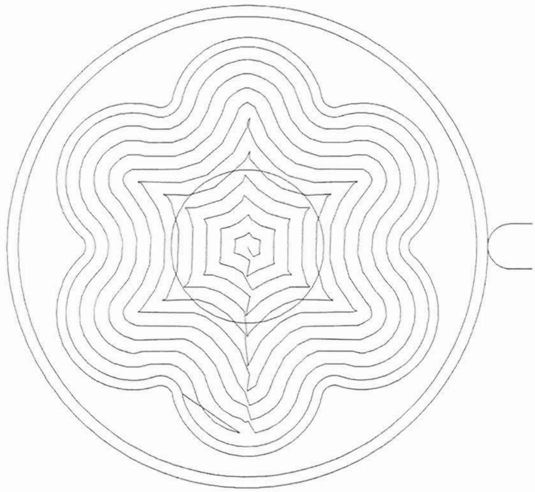
10. Select the **Roughing/Finishing parameters** tab.

11. Enter the values shown on the following dialog box.



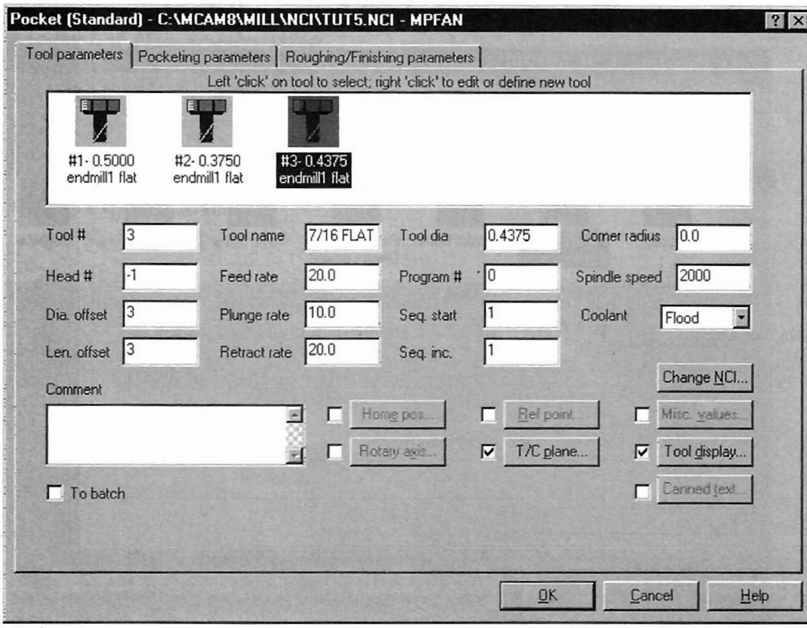
12. Choose **OK**.

The toolpath should look similar to the following picture.

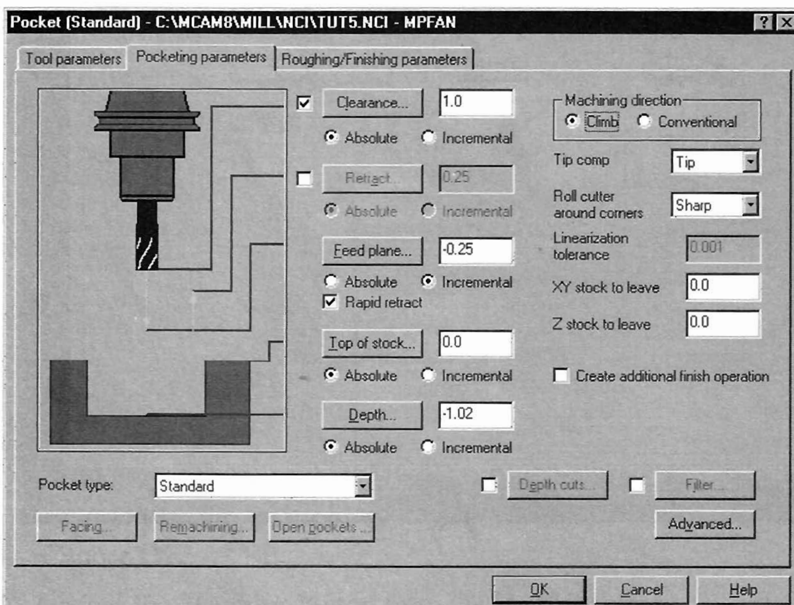


▶ *Pocket the inner bore*

1. Choose
   - ◆ **Pocket**
   - ◆ **Single**
2. Select the 5” diameter arc.
3. Choose **Done**.
4. Right-click in the tool display area and choose **Get tool from library**.
5. Select the 7/16” flat endmill.
6. Choose **OK**.
7. Enter the values shown on the following dialog box.

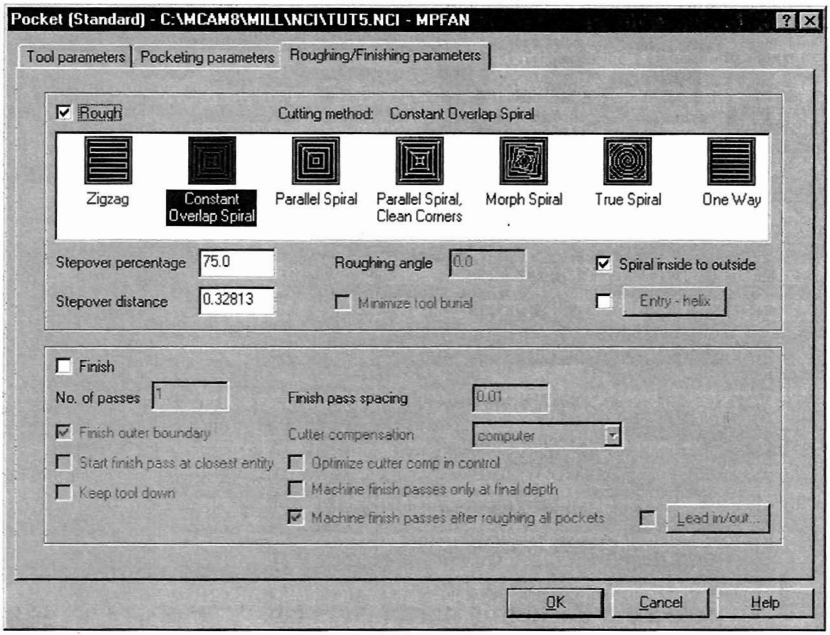


8. Select the **Pocketing parameters** tab.
9. Enter the values shown on the following dialog box.



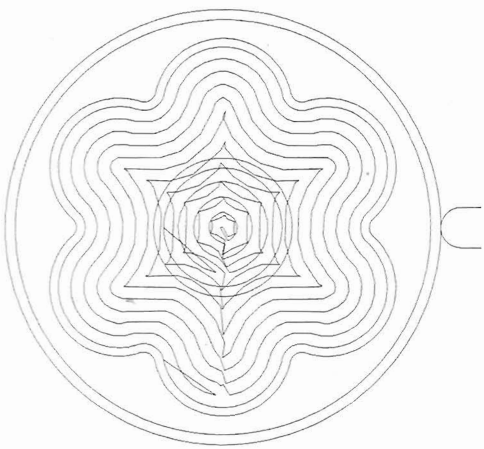
10. Select the **Roughing/Finishing parameters** tab.

11. Enter the values shown on the following dialog box.



12. Choose **OK**.

The toolpath should look like the following picture.



▶ *Backplot the toolpaths*

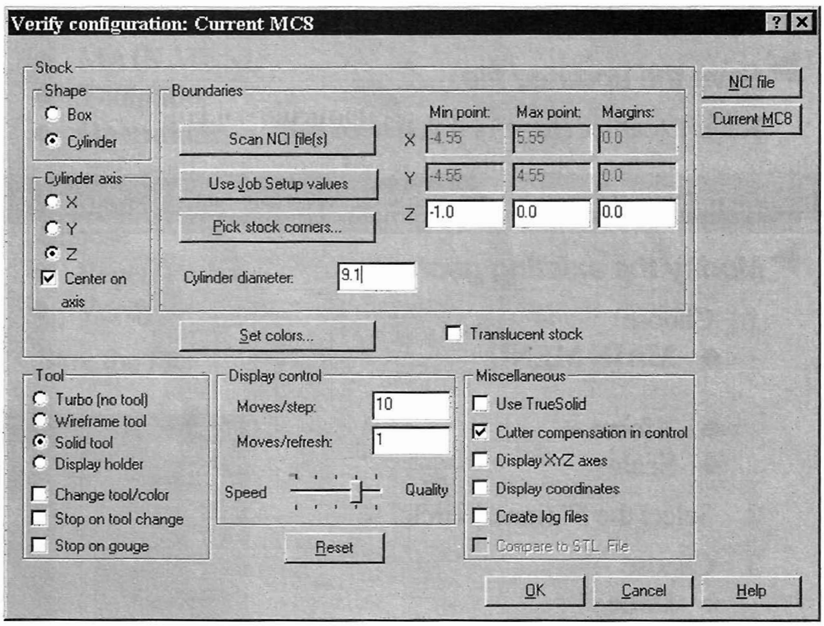
1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Toolpaths**
   - ◆ **Operations**
   - ◆ **Select All**
   - ◆ **Backplot**
   - ◆ **Run**

▶ *Verify the program*

1. Choose
   - ◆ **BACKUP**
   - ◆ **Verify**

2. Choose the **Configure** button from the toolbar.

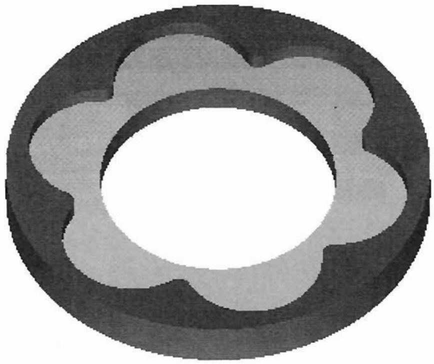
3. Enter the values shown on the following dialog box.



4. Choose **OK**.

5. Choose the **Machine** button from the Verify toolbar.

The part should look like the following picture.



6. Close the verify box.
7. Choose **OK** to exit the Operations Manager.

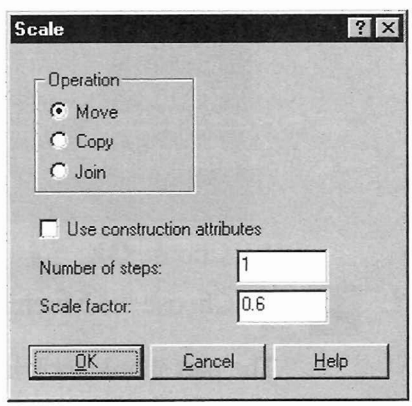
▶ *Save the updated file*

Refer to earlier chapters if you need help.

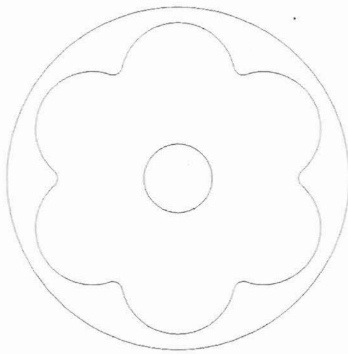
Step 3 – Associativity

▶ *Modify the existing geometry*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Xform**
   - ◆ **Scale**
2. Select the 5" (inner) circle.
3. Choose
   - ◆ **Done**
   - ◆ **Origin**
4. Enter the values shown at right.
5. Choose **OK**.



The changed geometry should look like the following picture.



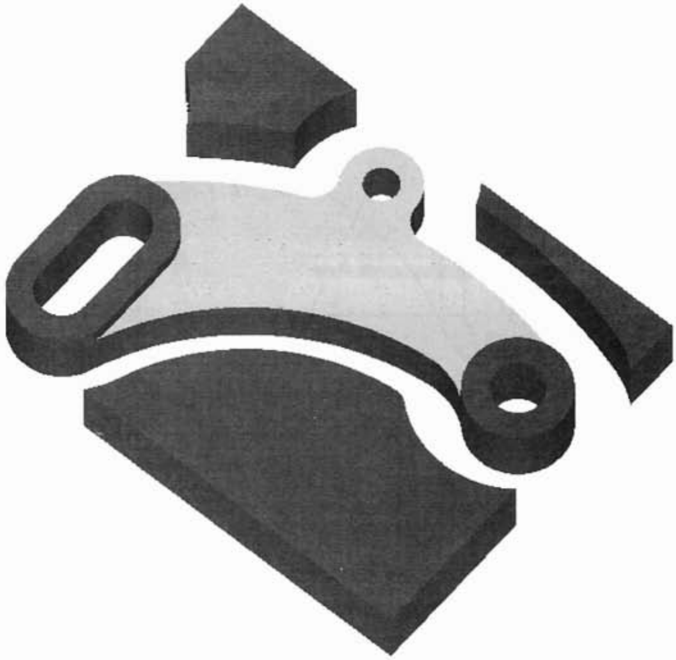
▶ *Update the toolpath*

1. Choose
   - **MAIN MENU**
   - **Toolpaths**
   - **Operations**

Note: Check marks should appear in all the folders.

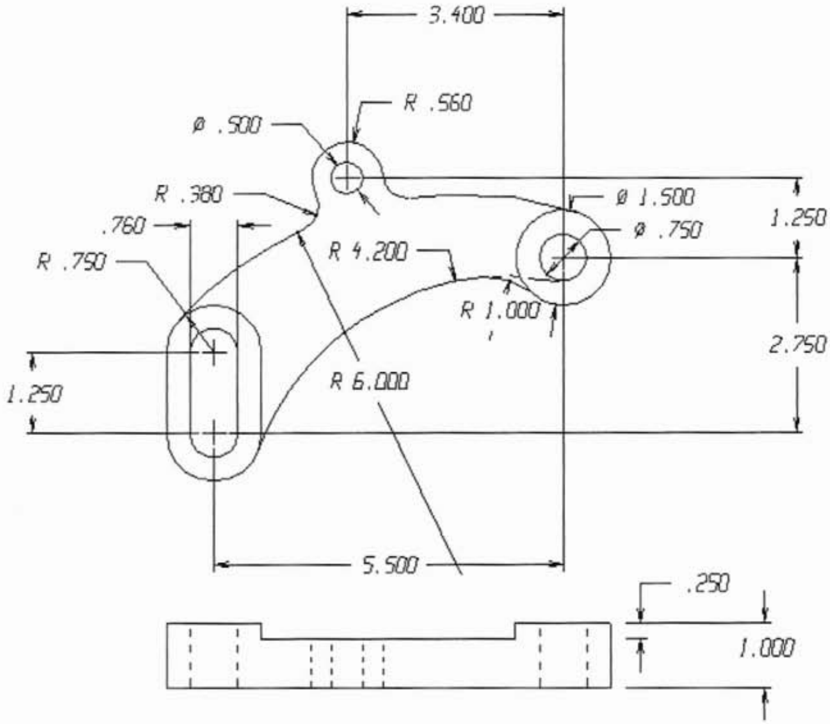
2. Choose
   - **Regen Path**
   - **Verify**
3. Save the file.

11 2D Pocket with Islands



Objectives

- Design a 3D wireframe drawing.
- Create arcs using polar positioning.
- Create lines using coordinate entry.
- Create arcs tangent to 2 existing entities.
- Trim 1 entity to another entity.
- Create parallel lines by defining the offset direction and distance.
- Break existing geometry to create a 3D block.
- Translate existing geometry to create a 3D block.
- Create 2D pocket and contour toolpaths.
- Perform solid model verification of the toolpath.



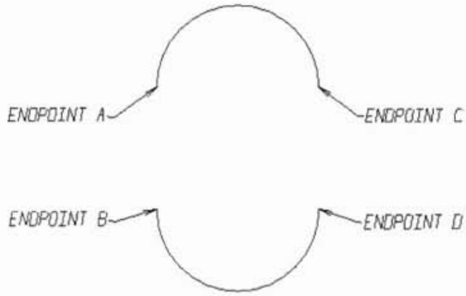
Geometry creation

▶ *Create the outside profile of the slot*

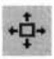
1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Arc**
   - ◆ **Polar**
   - ◆ **Center pt**
2. *Enter the center point.* **0, 0**
3. *Enter the radius.* **.75**
4. *Enter the initial angle.* **180**
5. *Enter the final angle.* **0**

6. *Enter the center point.* **0, 1.25**
7. *Enter the radius.* **.75**
8. *Enter the initial angle.* **0**
9. *Enter the final angle.* **180**

10. Choose
    - **MAIN MENU**
    - **Create**
    - **Line**
    - **Endpoints**
    - **Endpoint**



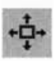
11. Select endpoint **A**.
12. Select endpoint **B**.
13. Select endpoint **C**.
14. Select endpoint **D**.



15. Choose the **Fit** button on the toolbar to center the geometry.

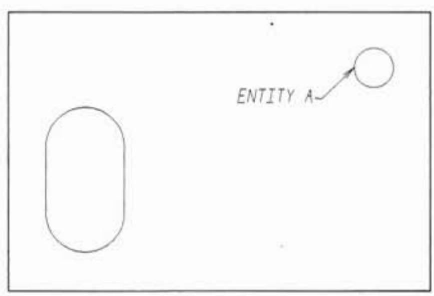
▶ *Create the 0.75 diameter and the 1.5 diameter arcs*

1. Choose
    - **MAIN MENU**
    - **Create**
    - **Arc**
    - **Circ pt+dia**
2. *Enter the diameter.* **.75**
3. *Enter the center point.* **5.5, 2.75**



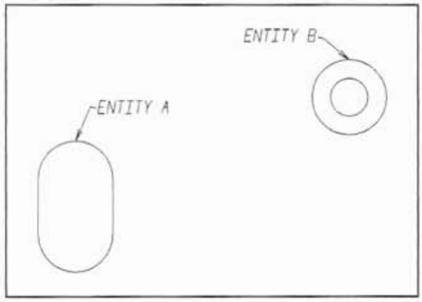
4. Choose the **Fit** button on the toolbar to center the geometry.

5. Choose
    - **BACKUP**
    - **Circ pt+dia**
6. *Enter the diameter.* **1.5**
7. Choose **Center**.
8. Select entity **A.**

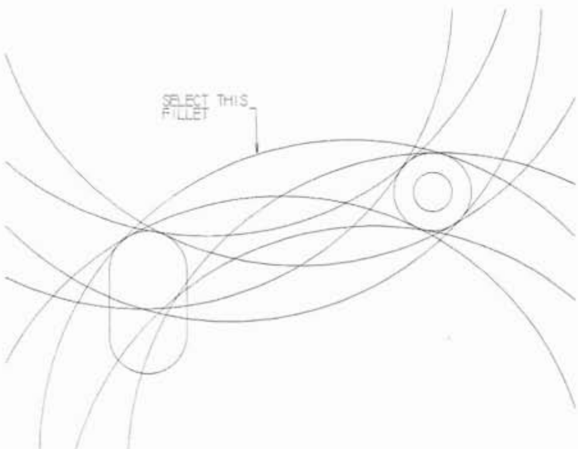


▶ *Create the 4.2 radius arc tangent to the preceding arcs*

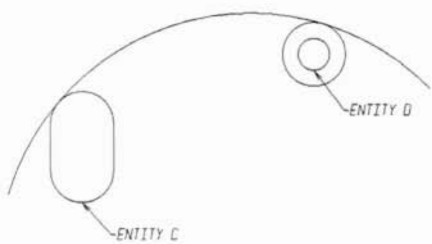
1. Choose
   - ◆ **BACKUP**
   - ◆ **Tangent**
   - ◆ **2 entities**
2. *Enter the radius*. **6**
3. Select entity **A** and entity **B** at the positions shown at right.



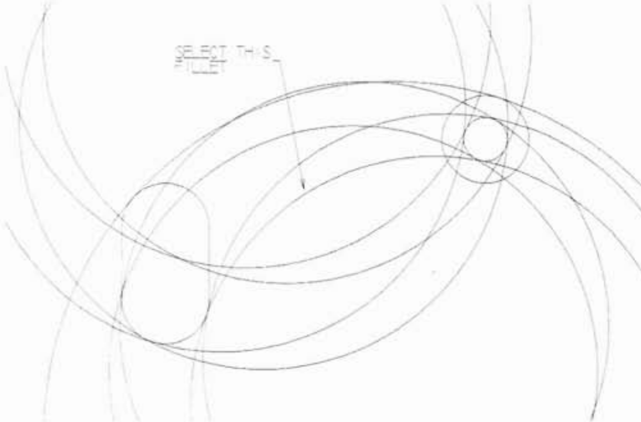
4. Select the fillet shown on the following picture.



5. Choose
   - ◆ **BACKUP**
   - ◆ **2 entities**
6. *Enter the radius*. **4.2**
7. Select entity **C** and entity **D** as shown.

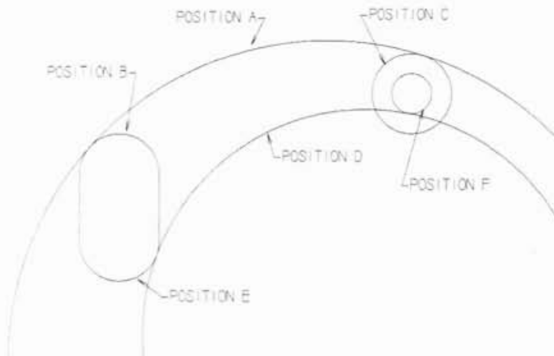


8. Select the fillet shown on the following drawing.



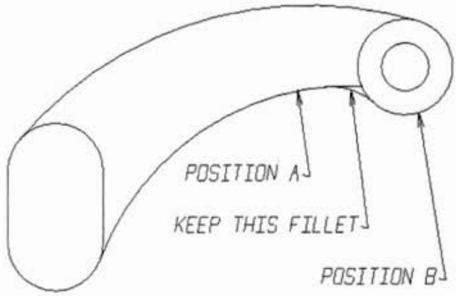
▶ *Trim the existing geometry*

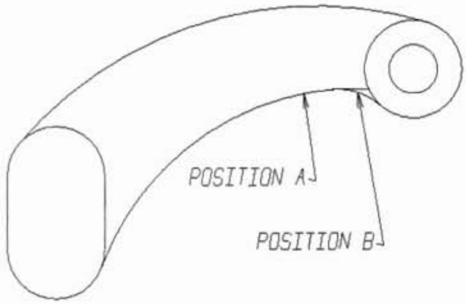
1. Choose the **Fit** button on the toolbar.
2. Choose
   - ◆ **MAIN MENU**
   - ◆ **Modify**
   - ◆ **Trim**
   - ◆ **1 entity**
3. Select position **A**.
4. Select position **B**.
5. Select position **A**.
6. Select position **C**.
7. Select position **D**.
8. Select position **E**.
9. Select position **D**.
10. Select position **F**.
11. Choose the **Fit** button on the toolbar.



Create the 1" radius fillet and trim the geometry

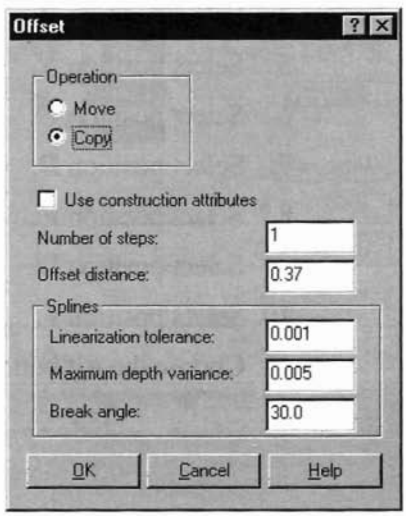
1. Choose
   - **BACKUP**
   - **BACKUP**
   - **Fillet**
   - **Radius**
2. *Enter the fillet radius.* **1**
3. Choose **Trim**. (Click on it to toggle this to **N**)
4. Select position **A**.
5. Select position **B**.
6. Select the fillet shown.
7. Choose
   - **MAIN MENU**
   - **Modify**
   - **Trim**
   - **1 entity**
8. Select position **A**.
9. Select position **B**.



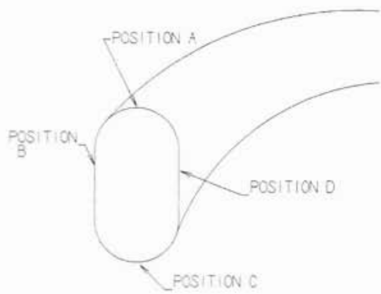


Offset the slot

1. Choose
   - **MAIN MENU**
   - **Xform**
   - **Offset**
2. Enter the values shown in the dialog box on the right.
3. Choose **OK**.

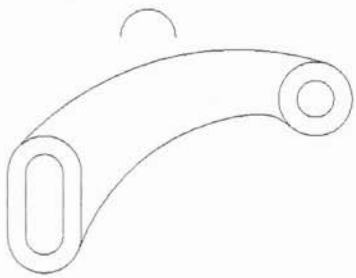


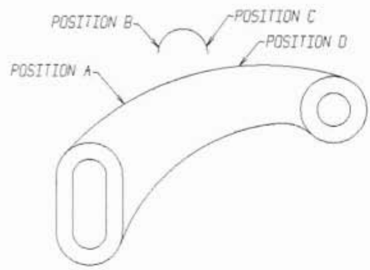
4. Select position **A**.
5. Select a point inside the slot.
6. Select position **B**.
7. Select a point inside the slot
8. Select position **C**.
9. Select a point inside the slot.
10. Select entity **D**.
11. Select a point inside the slot.



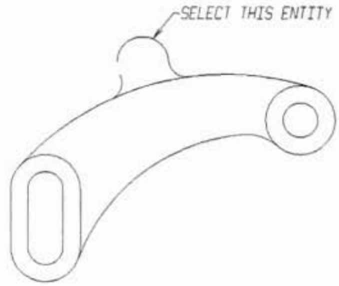
▶ *Finish creating the outside profile*

1. Choose
   - **MAIN MENU**
   - **Create**
   - **Arc**
   - **Polar**
   - **Center pt**
2. *Enter the center point.* **2.1, 4**
3. *Enter the radius.* **.56**
4. *Enter the initial angle.* **0**
5. *Enter the final angle.* **180**
6. Choose the **Fit** button on the toolbar.
7. Choose
   - **MAIN MENU**
   - **Create**
   - **Fillet**
   - **Radius**
8. *Enter the fillet radius.* **.380**
9. Select position **A**.
10. Select position **B**.
11. Select position **C**.
12. Select position **D**.

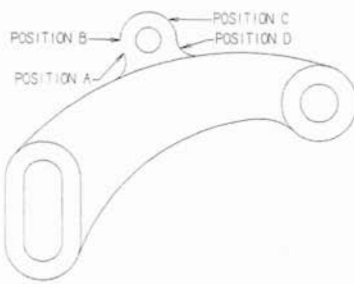




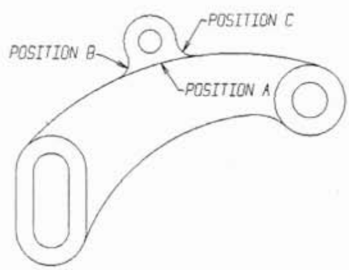
13. Choose
    - ◆ **BACKUP**
    - ◆ **Arc**
    - ◆ **Circ pt+dia**



14. *Enter the diameter.* **.5**
15. Choose **Center**.
16. Select the arc shown at right.
17. *Enter the diameter.* **.5**
18. Choose
    - ◆ **MAIN MENU**
    - ◆ **Modify**
    - ◆ **Trim**
    - ◆ **2 entities**

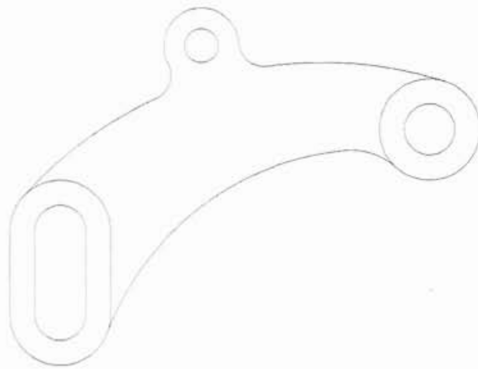


19. Select position **A**.
20. Select position **B**.
21. Select position **C**.
22. Select position **D**.
23. Choose
    - ◆ **BACKUP**
    - ◆ **Divide**



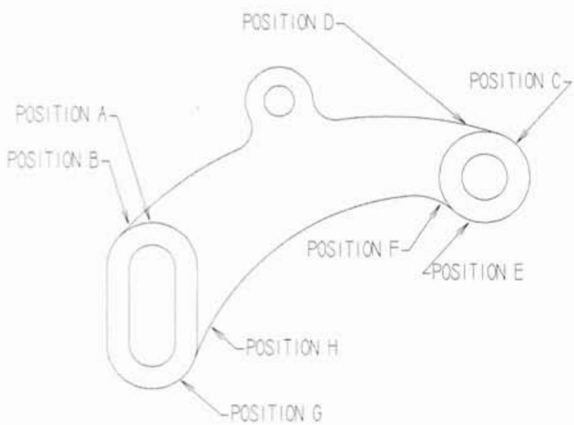
24. Select position **A**.
25. Select position **B**.
26. Select position **C**.
27. Choose
    - ◆ **MAIN MENU**
    - ◆ **Screen**
    - ◆ **Clr colors**

Your part should look like the following picture.



▶ *Break the existing arcs for future chaining*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Modify**
   - ◆ **Break**
   - ◆ **2 pieces**



2. Select position **A**.
3. Choose **Endpoint**.
4. Select position **B**.
5. Select position **C**.
6. Choose **Endpoint**.
7. Select position **D**.
8. Select position **E**.
9. Choose **Endpoint**.
10. Select position **F**.
11. Select position **G**.
12. Choose **Endpoint**.
13. Select position **H**.

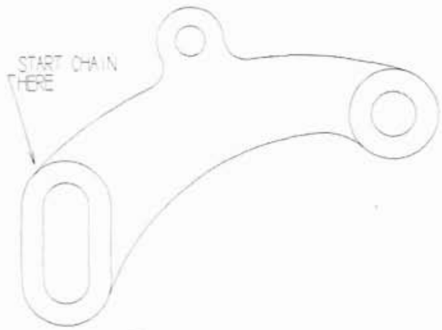
Save the file

1. Choose
   - **MAIN MENU**
   - **File**
   - **Save**
2. Enter your name followed by 11 and choose **Save**.

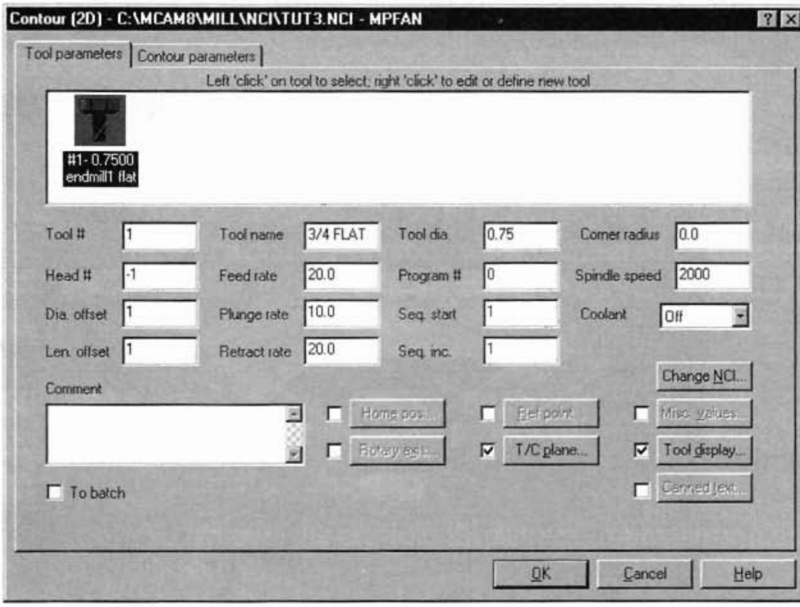
Toolpath creation

Create a contour around the part

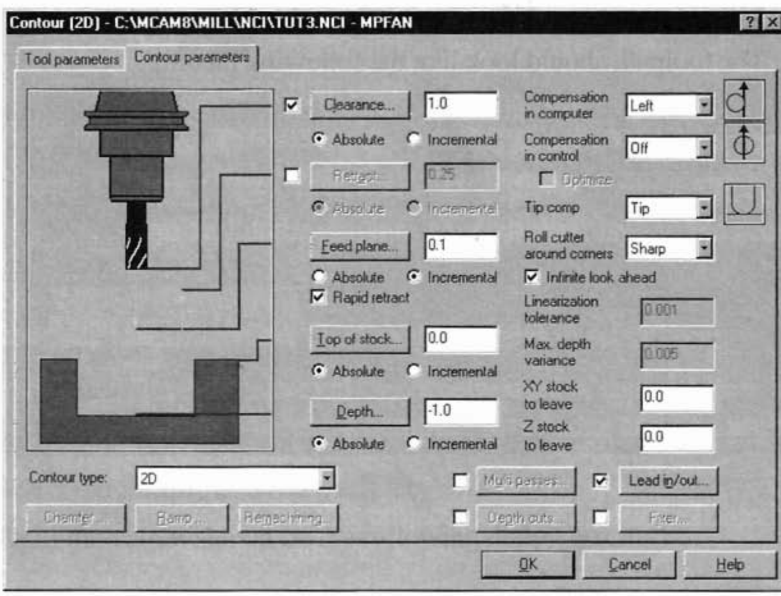
1. Choose
   - **MAIN MENU**
   - **Toolpaths**
   - **Contour**
2. Start the chain as shown in the following drawing. Use your mouse to guide the chain clockwise around the part by clicking on the next entity in the chain until you have gone all the way around the part.



2. Choose
   - **End here**
   - **Done**
3. Right-click in the tool display window and select **Get tool from library**.
4. Select the ¾" flat endmill and choose **OK**.
5. Enter the values shown on the following dialog box.

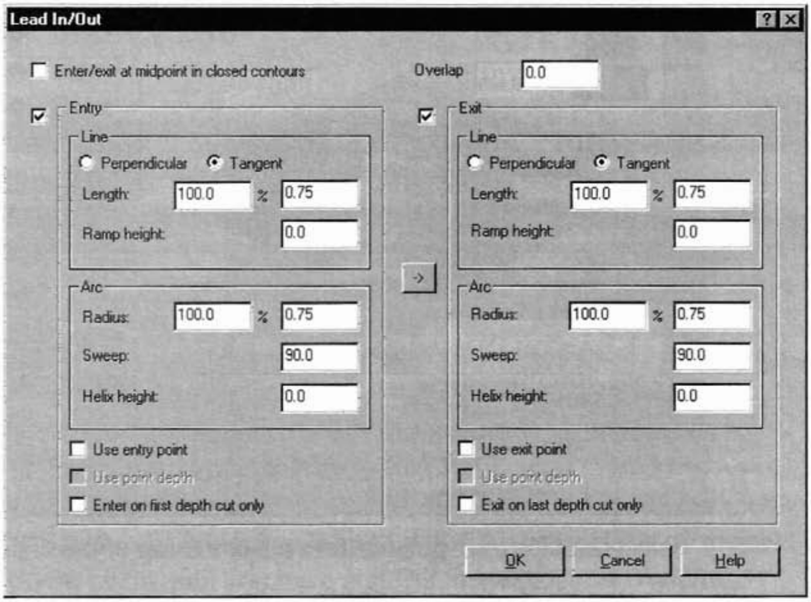


6. Select the **Contour parameters** tab at the top of the dialog box.
7. Enter the values shown in the following dialog box.



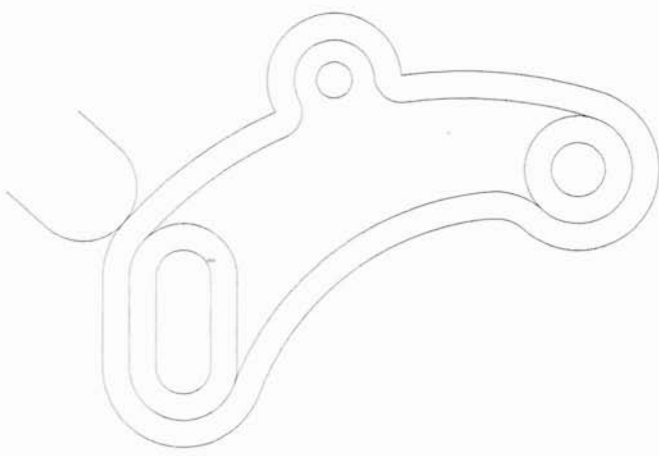
8. Select the **Lead in/out** check box and choose the **Lead in/out** button.

9. Enter the values shown in the following dialog box.



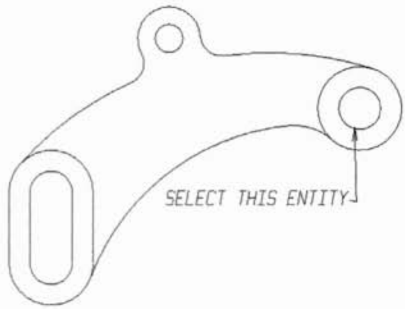
10. Choose **OK** twice.

The toolpath should look like the following picture.

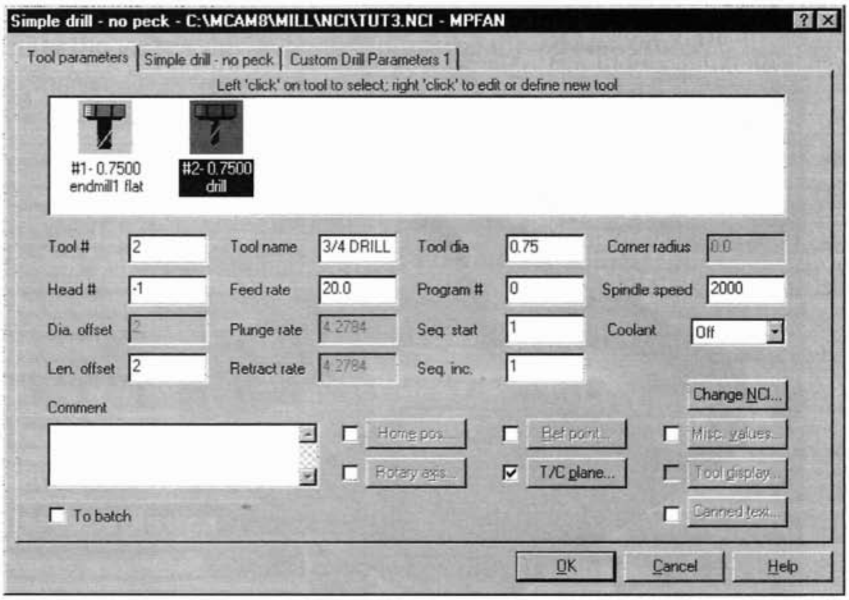


▶ *Drill 2 holes*

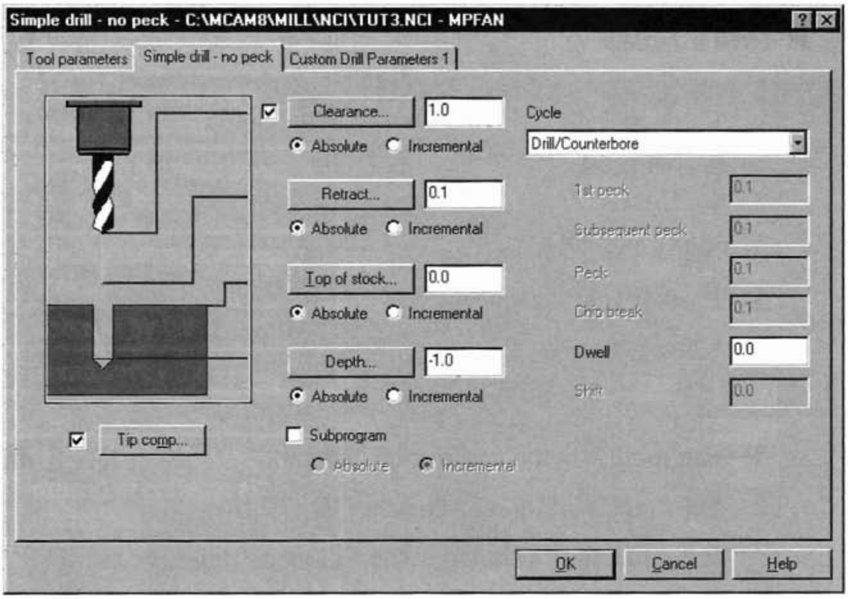
1. Choose
   - **Drill**
   - **Manual**
   - **Center**
2. Select the **.75** hole.
3. Press [Esc]
4. Choose **Done**.



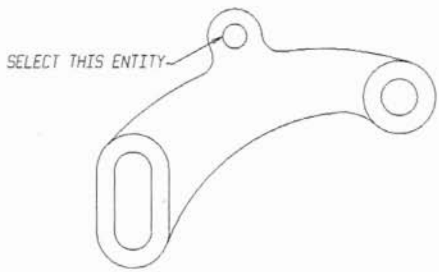
5. Right-click in the tool display area and select **Get tool from library**.
6. Select the ¾ drill" and choose **OK**.
7. Enter the values shown in the following dialog box.



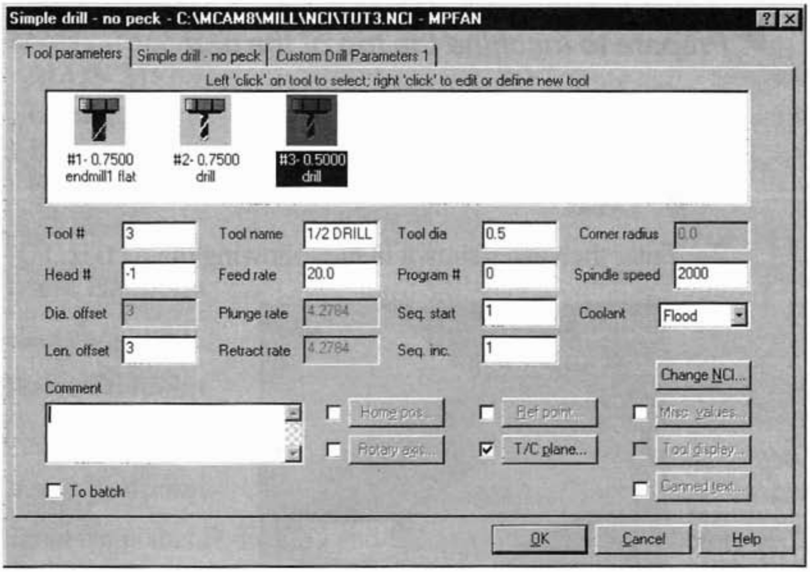
8. Select the **Simple drill/no peck** tab.
9. Enter the values shown on the following dialog box.



10. Choose **OK**.
11. Choose
    - **Drill**
    - **Manual**
    - **Center**
12. Select the **.5 diameter** hole.
13. Press [Esc].
14. Choose **Done**.

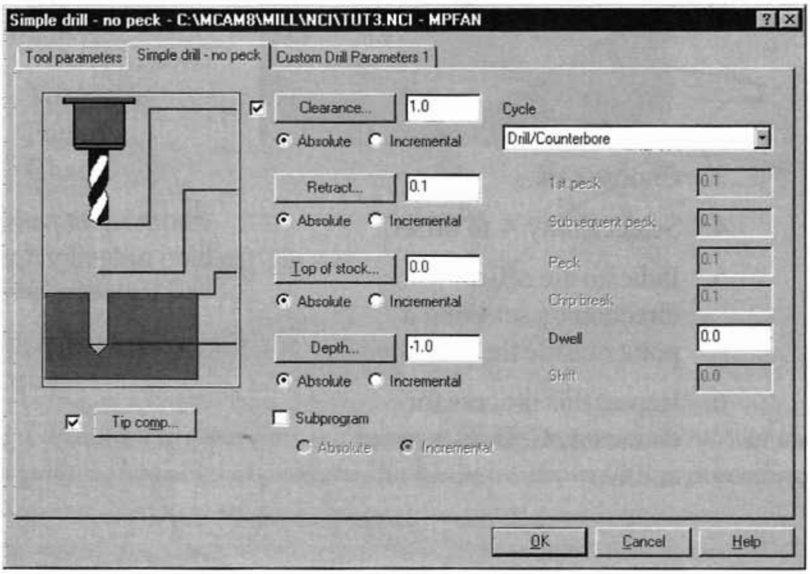


15. Right-click in the tool display area and select **Get tool from library**.
16. Select the ½ " drill and choose **OK**.
17. Enter the values shown on the following dialog box.



18. Select the **Simple drill/no peck** tab.

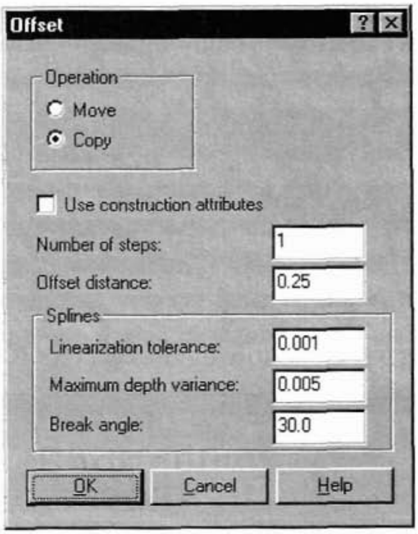
19. Enter the values shown on the following dialog box.



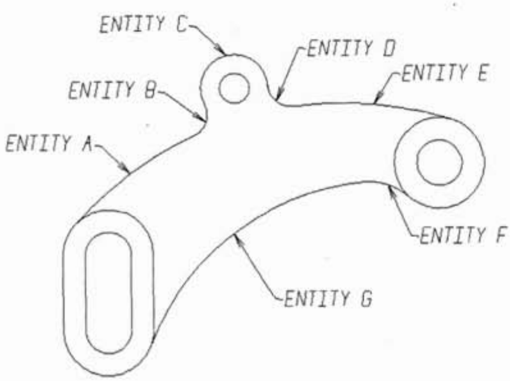
20. Choose **OK**.

▶ *Prepare to machine the top of the part*

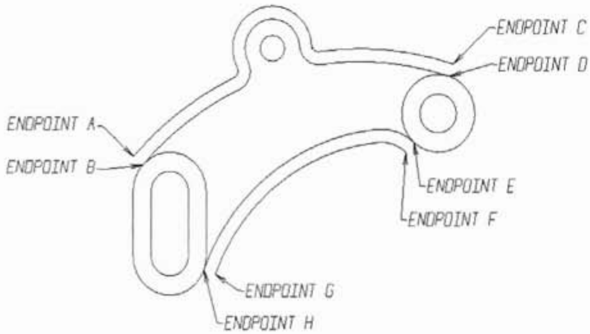
1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Xform**
   - ◆ **Offset**
2. Enter the values shown in the following dialog box.



3. Choose **OK**.
4. Select Entity **A** to offset.
5. Indicate the offset direction by selecting a point outside the part.
6. Repeat this process for entities **B**, **C**, **D**, **E**, **F**, and **G**.

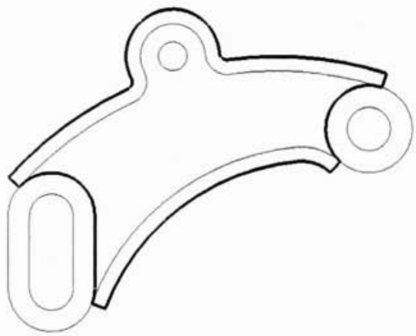


7. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Line**
   - ◆ **Endpoints**
8. Select endpoint **A**.
9. Choose **Endpoint**.
10. Select endpoint **B**.
11. Choose **Endpoint**.
12. Select endpoint **C**.
13. Choose **Endpoint**.
14. Repeat for points **D**, **E**, **F**, **G** and **H**, choosing **Endpoint** before each one.



▶ *Pocket the inside of the part*

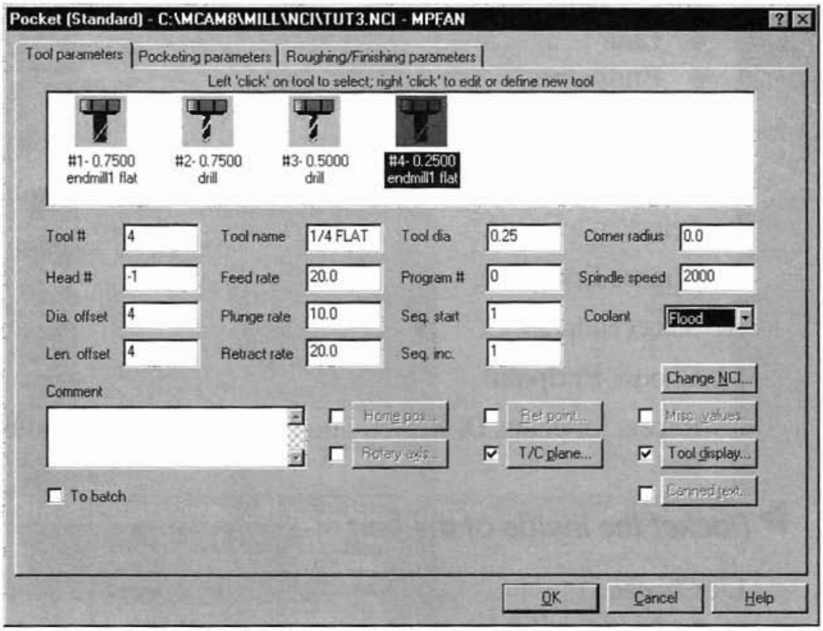
1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Toolpaths**
   - ◆ **Pocket**
   - ◆ **Chain**
2. Select the contour highlighted in bold at right to be pocketed.
3. Select the last entity in the contour.



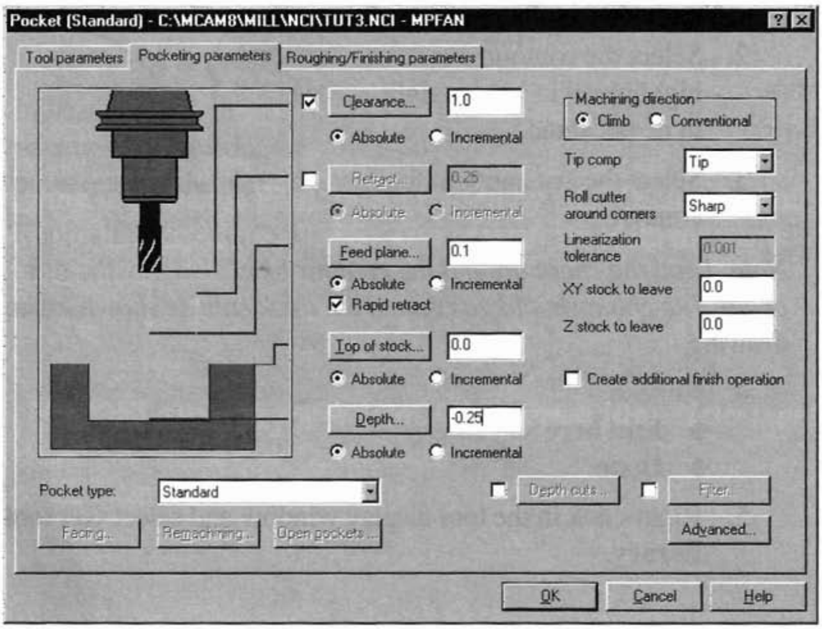
Note: Lead the chain around the contour by clicking on the next entity in the chain. The chain should go around the thick entities shown in the preceding drawing.

4. Choose
   - ◆ **End here**
   - ◆ **Done**
5. Right-click in the tool display window and select **Get tool from library**.

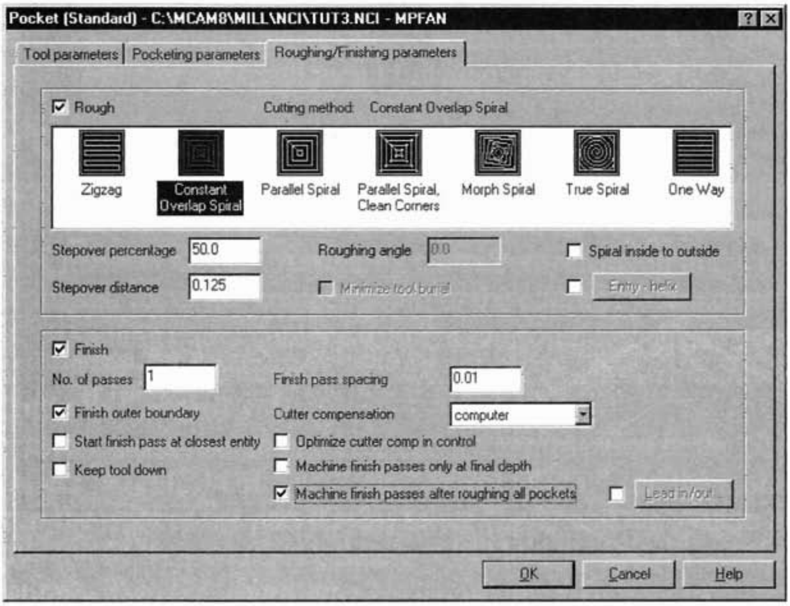
6. Select a ¼" flat endmill and choose **OK**.
7. Enter the values shown on the following dialog box.



8. Select the **Pocketing parameters** tab.
9. Enter the values shown on the following dialog box.



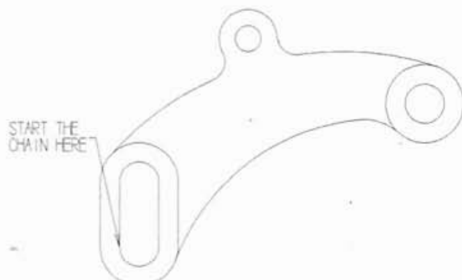
10. Select the **Roughing/Finishing parameters** tab.
11. Enter the values shown.



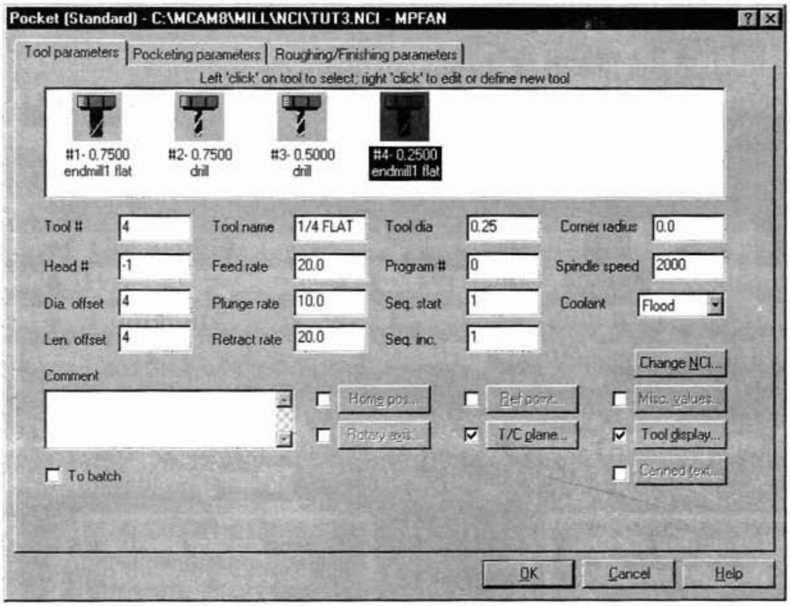
12. Choose **OK**.

► *Create a pocket toolpath for the 1.25 inch long slot*

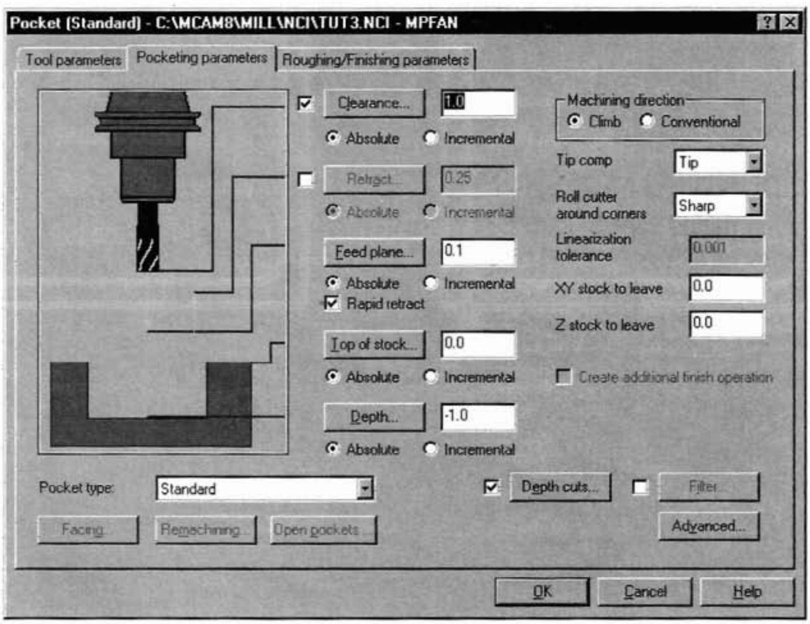
1. Choose
   - **Pocket**
   - **Chain**
2. Select the inside of the long arc.
3. Choose **Done**.



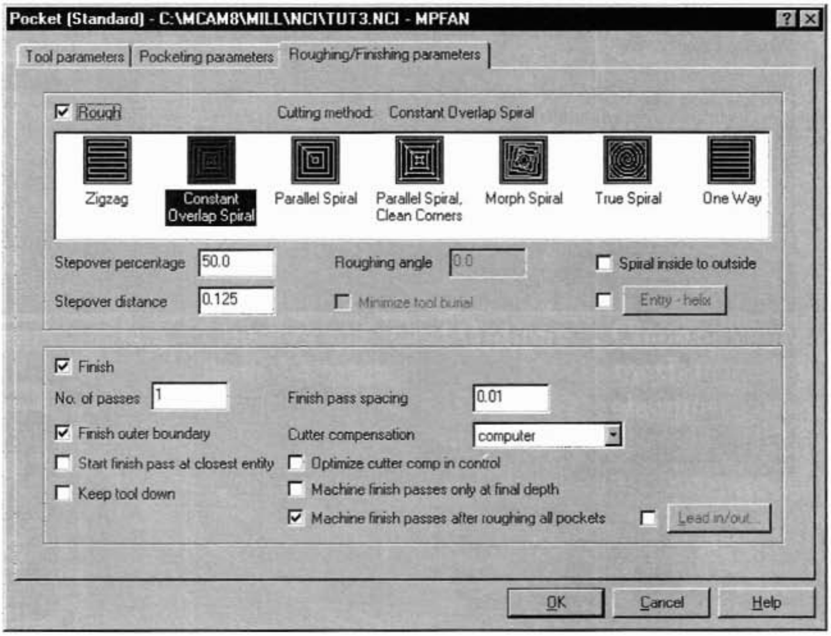
4. Select the ¼ " endmill used in the last operation.
5. Enter the values shown in the following dialog box.



6. Select the **Pocketing parameters** tab.
7. Enter the values shown on the following dialog box.
8. Select the check box next to the **Depth cuts** button.



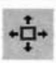
9. Select the **Roughing/Finishing parameters** tab.
10. Enter the values shown on the following dialog box.



11. Choose **OK**.

▶ *Backplot the toolpath*

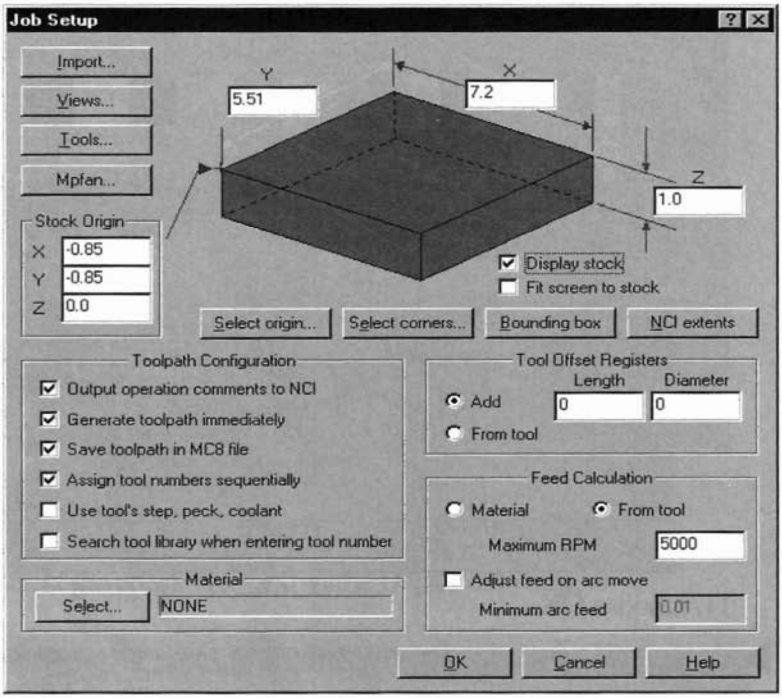
1. Choose the **Gview (isometric)** button from the toolbar



2. Choose the **Fit** button from the toolbar.
3. Choose
   - **MAIN MENU**
   - **Toolpaths**
   - **Operations**
   - **Select All**
   - **Backplot**
   - **Run**
4. Choose **BACKUP** to return to the Operations Manager.
5. Choose **OK**.

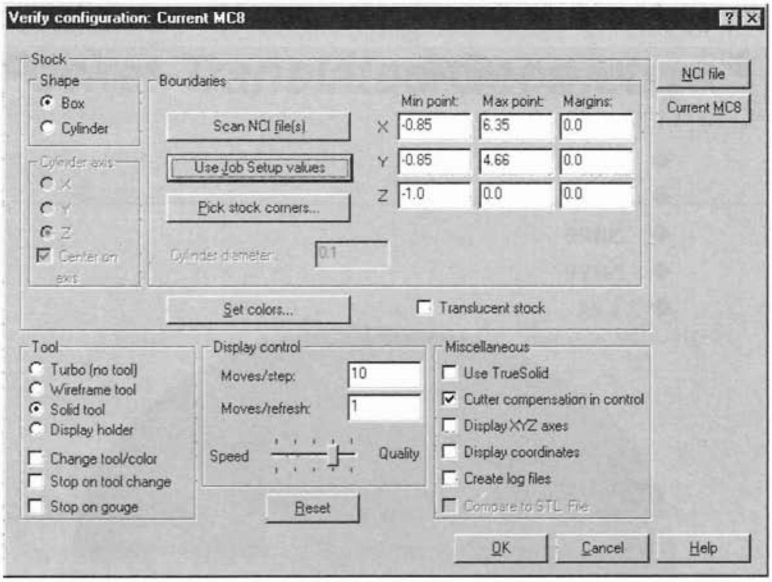
▶ *Verifying the toolpath*

1. Choose **Job setup**
2. Enter the values shown on the following dialog box.



3. Choose **OK**.
4. Choose **Operations**.
5. Choose
   - **Select All**
   - **Verify**

6. Choose the **Configure** button from the toolbar.
7. Select the **Use Job Setup** values button.
8. Enter the values shown on the following dialog box.



9. Choose **OK**.

10. Choose the **Machine** button from the Verify toolbar.

The part should look like the following picture.

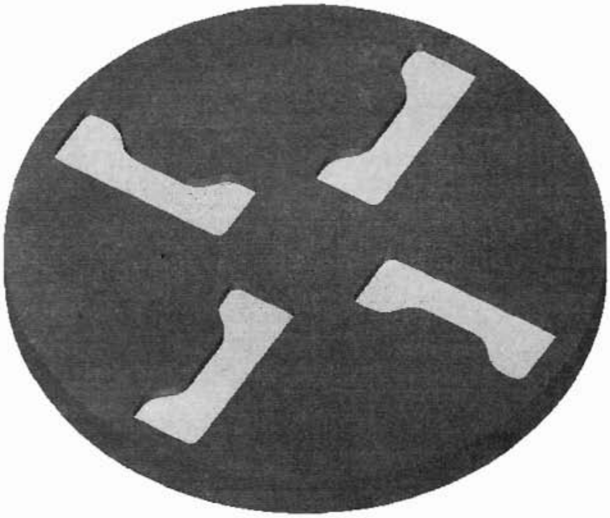
11. Close the Verify toolbar.

12. Choose **OK** to exit the Operations Manager.

▶ *Save the updated file*

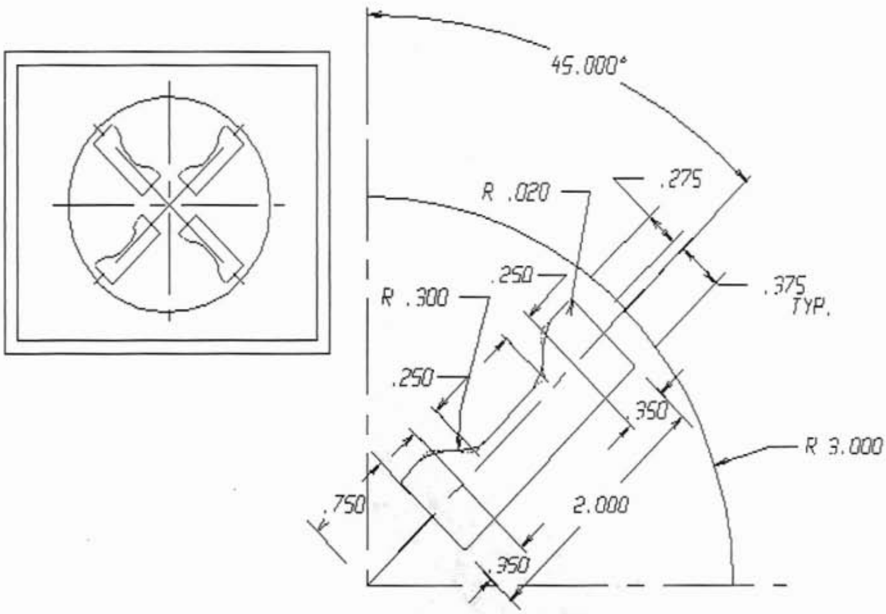
1. Choose
    - **MAIN MENU**
    - **File**
    - **Save**
    - **Save**
    - **Yes**

12 Pocket-Translated Geometry



Objectives

- Design a 2D drawing.
- Create arcs and lines using polar positioning.
- Create perpendicular lines to existing geometry.
- Create parallel lines by defining the offset direction and distance.
- Break existing geometry into two sections.
- Create fillet radii.
- Mirror existing geometry to complete a part.
- Edit an existing toolpath to complete the machining operations.
- Perform solid model verification of the toolpath.



Geometry creation

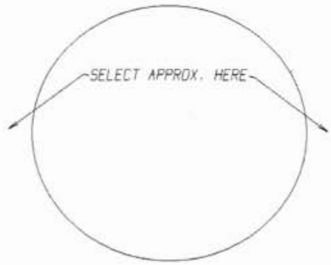
▶ *Create the outside diameter*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Arc**
   - ◆ **Circ pt+rad**
2. *Enter the radius.* **3**
3. *Enter the coordinates.* **0,0**

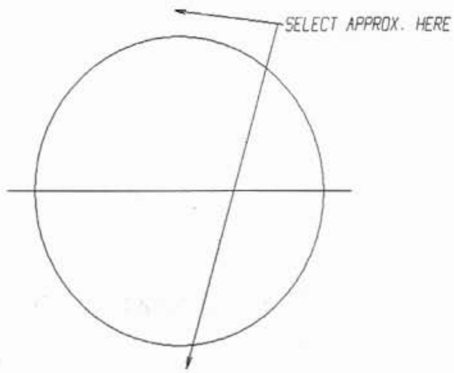
▶ *Create the center lines*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Line**
   - ◆ **Horizontal**

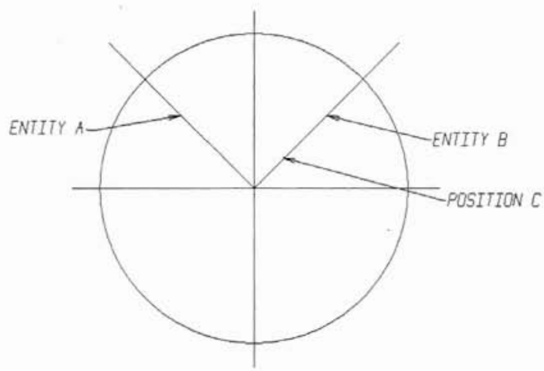
2. Select two points outside the circle at approximately 0 and 180 degrees as shown at right.
3. *Enter the Y coordinate.* **0**
4. Choose
   - ◆ **BACKUP**
   - ◆ **Vertical**



5. Select two points outside the circle at approximately 90 and 270 degrees as shown in the following picture.
6. *Enter the X coordinate.* **0**
7. Choose
   - ◆ **BACKUP**
   - ◆ **Polar**
   - ◆ **Center**
8. Select the circle.
9. *Enter the angle.* **45**
10. *Enter the line length.* **4**

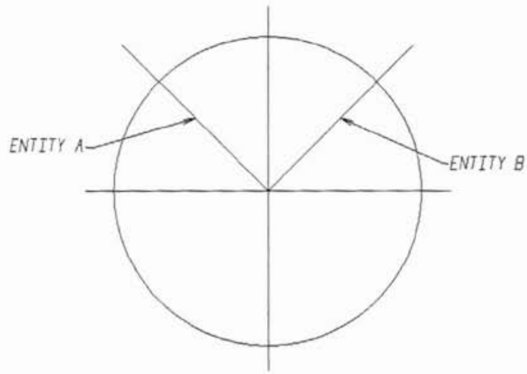


11. Choose
    - ◆ **MAIN MENU**
    - ◆ **Create**
    - ◆ **Line**
    - ◆ **Perpendclr**
    - ◆ **Point**
12. Select entity **B**.
13. Choose **Endpoint**.
14. Select position **C**.
15. Enter the line length. **4**
16. Select entity **A** as the line to keep.



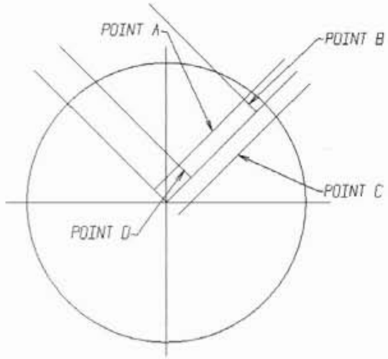
▶ *Create a rectangle in the proper orientation*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Line**
   - ◆ **Parallel**
   - ◆ **Side/dist**



2. Select entity **A**.
3. Select a point to the right of the line.
4. *Enter the parallel line distance.* **.75**
5. Select the new line.
6. Select a point to the right of the new line.
7. *Enter the parallel line distance.* **2**
8. Select entity **B**.
9. Select a point above the line.
10. *Enter the parallel line distance.* **.375**
11. Select entity **B** again.
12. Select a point below the line.
13. *Enter the parallel line distance.* **.375**
14. Choose
    - ◆ **MAIN MENU**
    - ◆ **Modify**
    - ◆ **Fillet**
    - ◆ **Radius**
15. *Enter the fillet radius.* **.02**

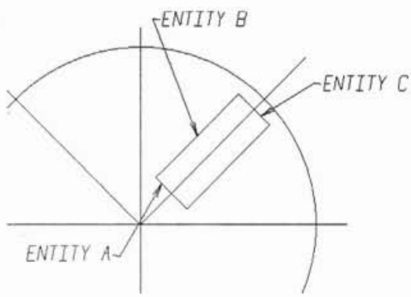
16. Select point **A**.
17. Select point **B**.
18. Select point **B** again.
19. Select point **C**.
20. Select point **C** again.
21. Select point **D**.
22. Select point **D** again.
23. Select point **A**.



▶ *Create lines to define the rough inside shape*

Note: Zoom in on the new rectangle in the top-right corner by clicking the Zoom button on the toolbar. Place a window around the area.

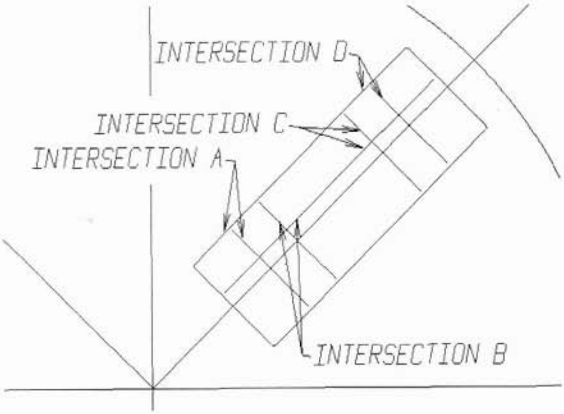
1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Line**
   - ◆ **Parallel**
   - ◆ **Side/dist**
2. Select entity **A**.
3. Select a point to the right of the line.
4. *Enter the parallel line distance.* **.350**
5. Select the new line.
6. Select a point to the right of the line.
7. *Enter the parallel line distance.* **.250**
8. Select entity **C**.
9. Select a point to the left of the line.
10. *Enter the parallel line distance.* **.350**
11. Select the new line.
12. Select a point to the left of the line.
13. *Enter the parallel line distance.* **.250**
14. Select entity **B**.



15. Select a point below the line.
16. *Enter the parallel line distance.* **.275**
17. Choose
    - **BACKUP**
    - **BACKUP**
    - **Endpoints**

Note: Choose two intersection lines for each intersection.

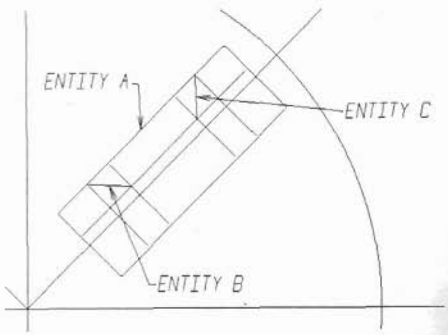
18. Choose **Intersect**.
19. Select each of the two intersecting lines at **A**.
20. Choose **Intersect**.
21. Select intersecting lines at **B**.
22. Choose **Intersect**.



23. Select intersecting lines at **C**.
24. Choose **Intersect**.
25. Select intersecting lines at **D**.

▶ *Trim the shape*

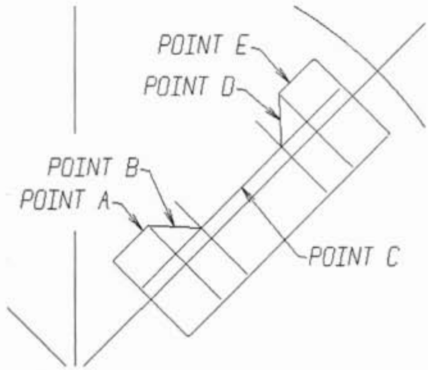
1. Choose
    - **MAIN MENU**
    - **Modify**
    - **Trim**
    - **Divide**
2. Select entity **A** as the curve to divide.
3. Select entity **B** as the first dividing curve.



4. Select entity **C** as the second dividing curve.

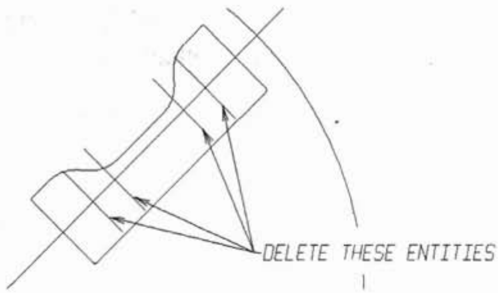
▶ Fillet and trim the shape

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Modify**
   - ◆ **Fillet**
   - ◆ **Radius**
2. *Enter the fillet radius.* **.3**
3. Select point **A**.
4. Select point **B**.
5. Select point **B**.
6. Select point **C**.
7. Select point **C**.
8. Select point **D**.
9. Select point **D**.
10. Select point **E**.



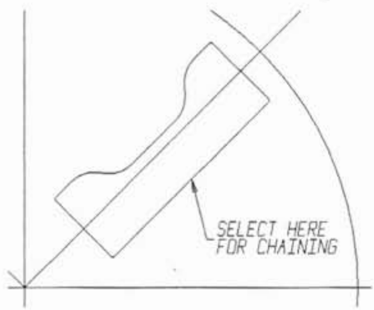
▶ Delete the construction lines

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Delete**
2. Select the entities shown.

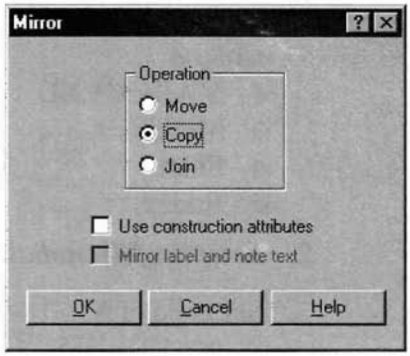


▶ Mirror the part

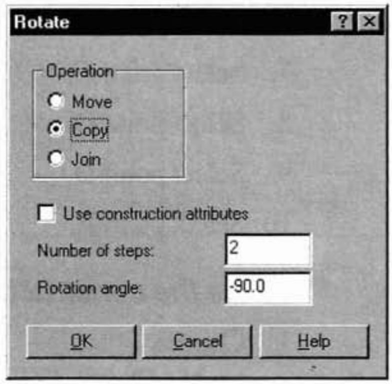
1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Xform**
   - ◆ **Mirror**
   - ◆ **Chain**
2. Select the entity as shown at right.
3. Choose
   - ◆ **Done**
   - ◆ **Done**
   - ◆ **X axis**



4. Select **Copy** in the dialog box at right.



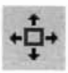
5. Choose **OK**.
6. **Fit** the part to the screen.
7. Choose
   - **BACKUP**
   - **Rotate**
   - **Result**
   - **Done**
   - **Origin**
8. Enter the values shown in the following dialog box.



9. Choose **OK**.

10. Choose **Clear colors** button from the toolbar.

11. Choose the **Unzoom** button from the toolbar.



12. Choose the **Fit** button from the toolbar.

▶ *Save the geometry*

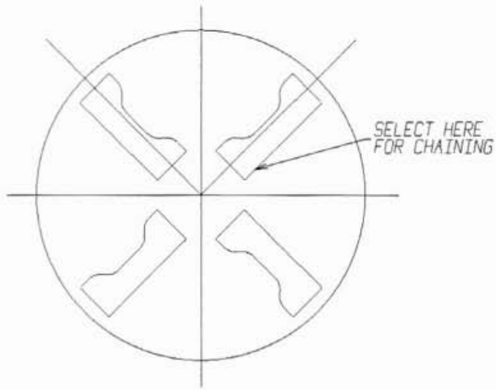
1. Choose
   - **MAIN MENU**
   - **File**
   - **Save**
2. Save this file using your name followed by 12.

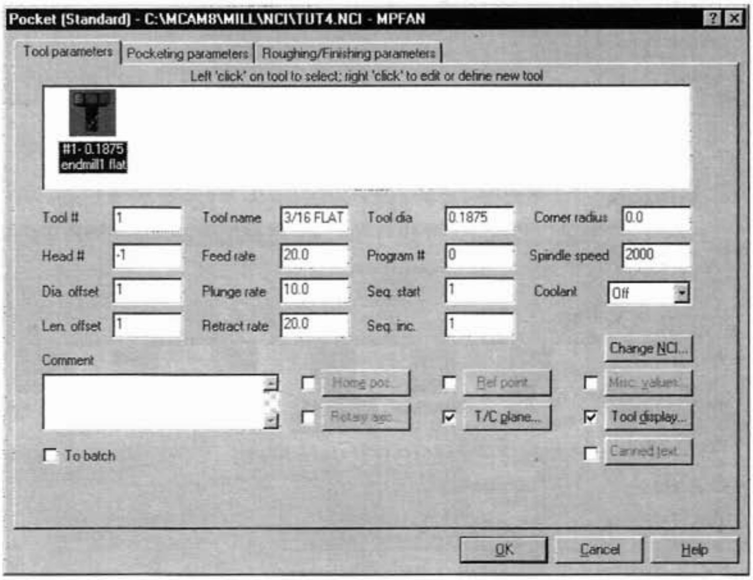
Step 2 - Toolpath creation

▶ *Create the pocket toolpath for the original geometry*

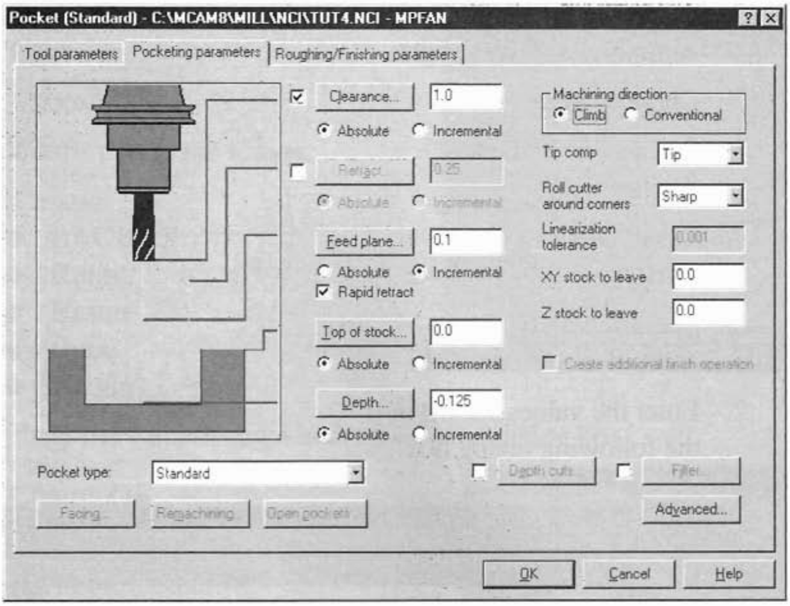
1. Choose
   - **MAIN MENU**
   - **Toolpaths**
   - **Pocket**
   - **Chain**

2. Select the first entity in the contour as shown at right.
3. Choose **Done**.
4. Right-click in the tool display area.
5. Select **Get tool from library**.
6. Select the 3/16" flat endmill and choose **OK**.
7. Enter the values shown in the following dialog box.



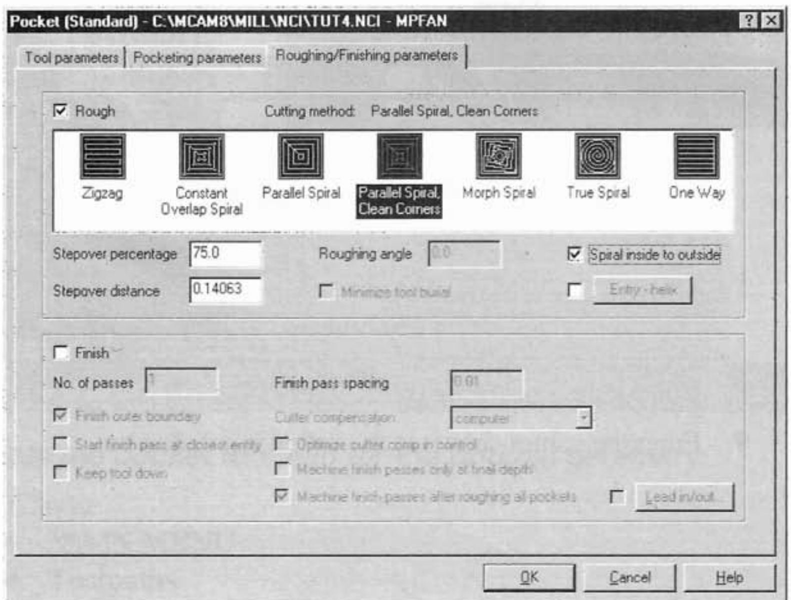


8. Choose the **Pocketing parameters** tab.
9. Enter the values shown in the following dialog box.



10. Select the **Roughing/Finishing parameters** tab.

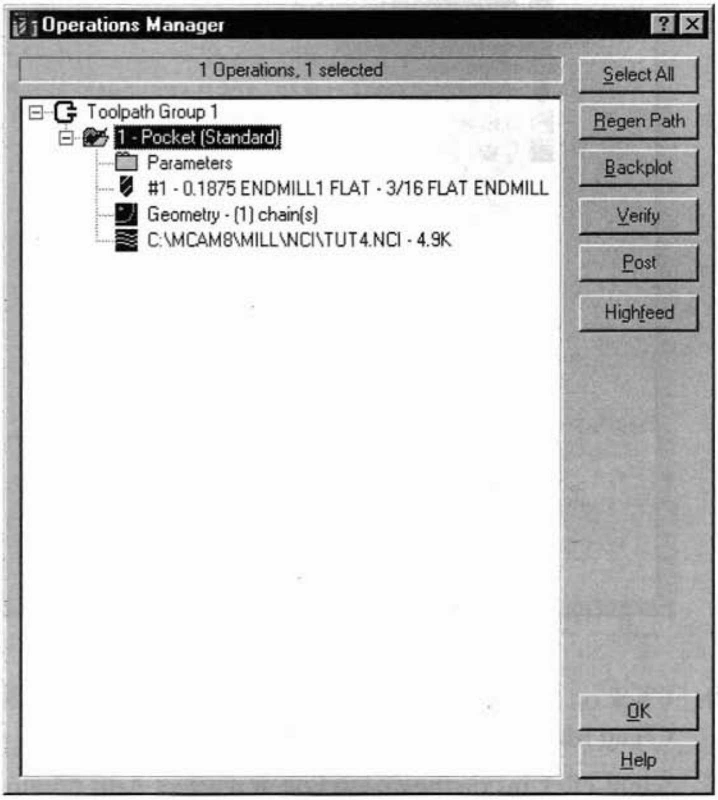
11. Enter the values shown in the following dialog box



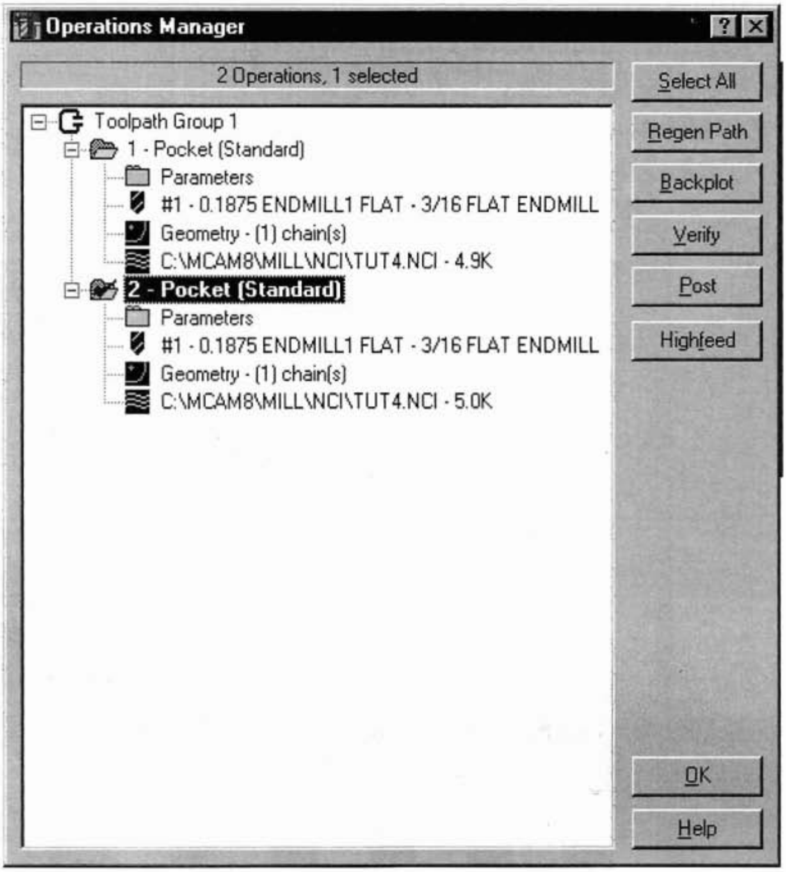
12. Choose **OK**.

▶ *Transform the toolpath to complete the remaining pockets*

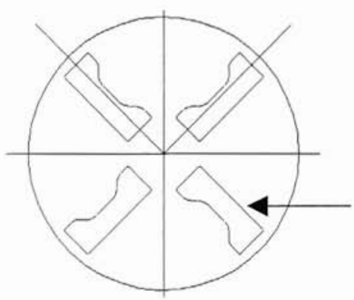
1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Toolpaths**
   - ◆ **Operations**



2. Right-click on the Pocket folder and drag it to the end of the list.
3. Choose **Copy After**. This creates a second pocket toolpath as shown in the following picture.



4. Click on the second **Geometry** icon in the Operations Manager. The Chain Manager dialog box opens.
5. Right-click inside the dialog box and select **Add Chain**.
6. Select the next pocket as shown in the following picture.



7. Choose **Done**. The Chain Manager should now show two chains.

8. Click once on Chain 1 and press [Delete].
9. Choose **OK**.
10. Repeat steps 2 through 10 for the remaining two pockets.

The Operations Manager should now show 4 pocket operations.

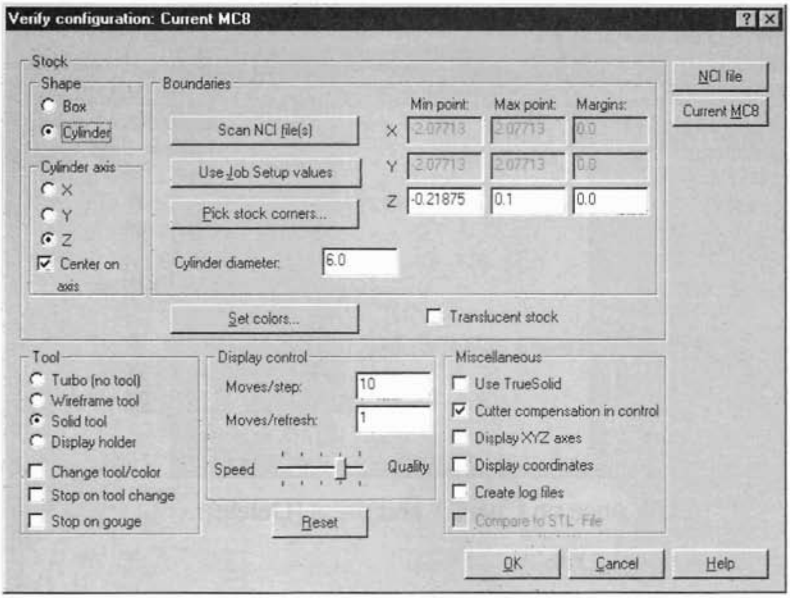
▶ *Backplot the toolpath*

1. Choose
   - ◆ **Select All**
   - ◆ **Regen Path**
   - ◆ **Backplot**
2. Change the Gview to **Isometric**.
3. Choose
   - ◆ **Run**
   - ◆ **BACKUP**

▶ *Verify the toolpath*

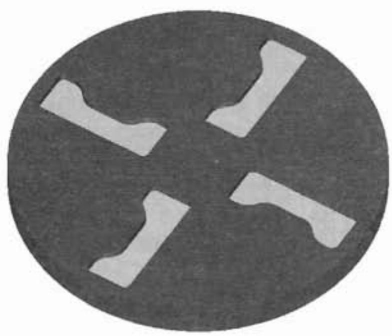
1. Choose **Verify**.

2. Choose the **Configure** button from the toolbar.
3. Enter the values shown on the following dialog box.



5. Choose
   - ◆ **OK**
   - ◆ **Machine**

The verified part should look like the following picture.

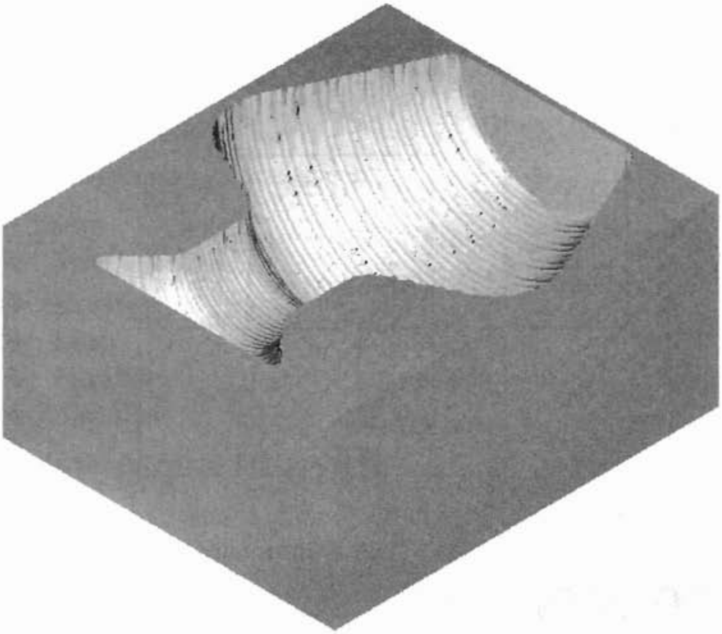


6. Close the Verify toolbar.
7. Choose **OK** to exit the Operations Manager.

▶ *Save the updated file*

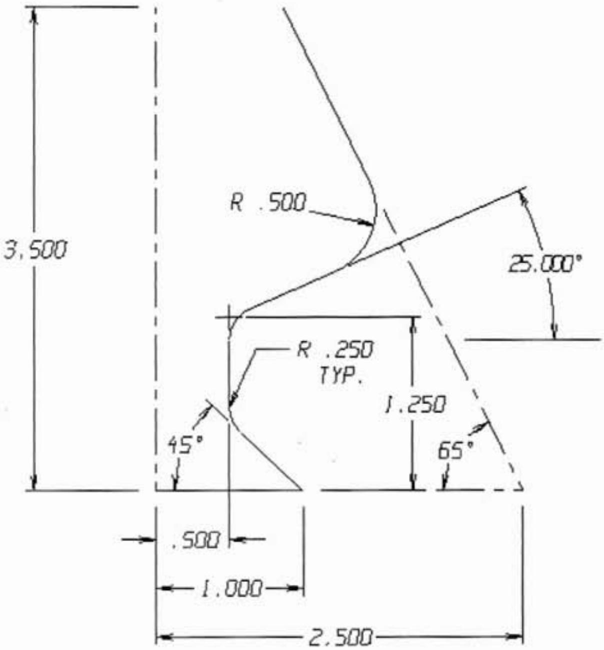
Refer to earlier chapters if you need help.

13 3D Revolved Surface - Parallel



Objectives

- Design a 3D drawing.
- Create lines using polar positioning.
- Create parallel lines by defining the offset direction and distance.
- Create a revolved surface.
- Change view for better visualization.
- Create a 3D rough and 3D finish toolpath.
- Define a 3D block the size of the workpiece.
- Perform solid model verification of the toolpath.



Geometry creation

▶ *Create the profile*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Line**
   - ◆ **Polar**
   - ◆ **Origin**
2. *Enter the angle in degrees.* **90**
3. *Enter the line length.* **3.5**
4. Choose **Origin**.
5. *Enter the angle in degrees.* **0**
6. *Enter the line length.* **2.5**
7. *Specify an endpoint.* **2.5, 0**

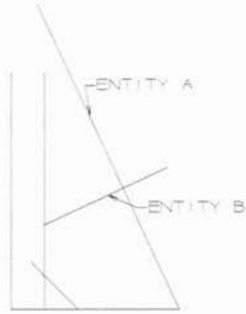
8. *Enter the angle in degrees.* **180-65**
9. *Enter the line length.* **5**
10. *Specify an endpoint.* **1, 0**
11. *Enter the angle in degrees.* **180-45**
12. *Enter the line length.* **1**
13. *Specify an endpoint.* **0.5, 1.25**
14. *Enter the angle in degrees.* **25**
15. *Enter the line length.* **2**
16. Fit the geometry to the screen.
17. Choose
    - **MAIN MENU**
    - **Create**
    - **Line**
    - **Parallel**
    - **Side/dist**
18. Select the vertical line to the far left.
19. Select a point to the right of that line.

20. *Enter the parallel distance.* **.5**

Your geometry should look like the following picture.

▶ Create fillets and trim to complete the profile

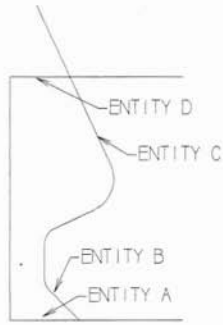
1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Fillet**
   - ◆ **Radius**



2. *Enter the fillet radius.* **.5**
3. Select entity **A**.
4. Select entity **B**.
5. Choose **Radius**.
6. *Enter the fillet radius.* **.25**
7. Select entity **1**.
8. Select entity **2**.
9. Select entity **2**.
10. Select entity **3**.

11. Choose
    - ◆ **MAIN MENU**
    - ◆ **Create**
    - ◆ **Line**
    - ◆ **Parallel**
    - ◆ **Side/dist**
12. Select the bottom horizontal line as shown in the following picture.

13. Select above that line.
14. *Enter the parallel line distance.* **3.5**
15. Choose
    - ◆ **MAIN MENU**
    - ◆ **Modify**
    - ◆ **Trim**
    - ◆ **1 entity**
16. Select entity **A** as the entity to trim.
17. Select entity **B** as the entity to trim to.
18. Select entity **C** as the entity to trim.
19. Select entity **D** as the entity to trim to.

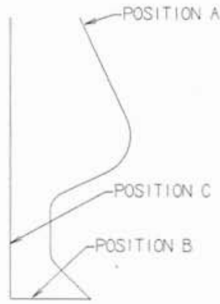


▶ *Delete the construction line*

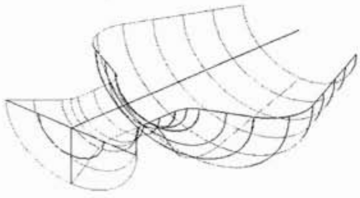
1. Choose
    - ◆ **MAIN MENU**
    - ◆ **Delete**
2. Select the construction line (the top horizontal line).

▶ *Create the surface*

1. Choose
    - ◆ **MAIN MENU**
    - ◆ **Create**
    - ◆ **Surface**
    - ◆ **Revolve**
    - ◆ **Chain**
    - ◆ **Partial**
2. Select position **A**.
3. Select position **B**.
4. Choose **Done**.
5. Select position **C**.
6. Choose **Start angle**.
7. *Enter the start angle.* **0**
8. Choose **End angle**.



9. *Enter the end angle.* **180**
10. Choose **Do it**.
11. Change the Gview to **Isometric**.
12. Fit the geometry to the screen. The part should look like the following picture.



▶ *Save the file*

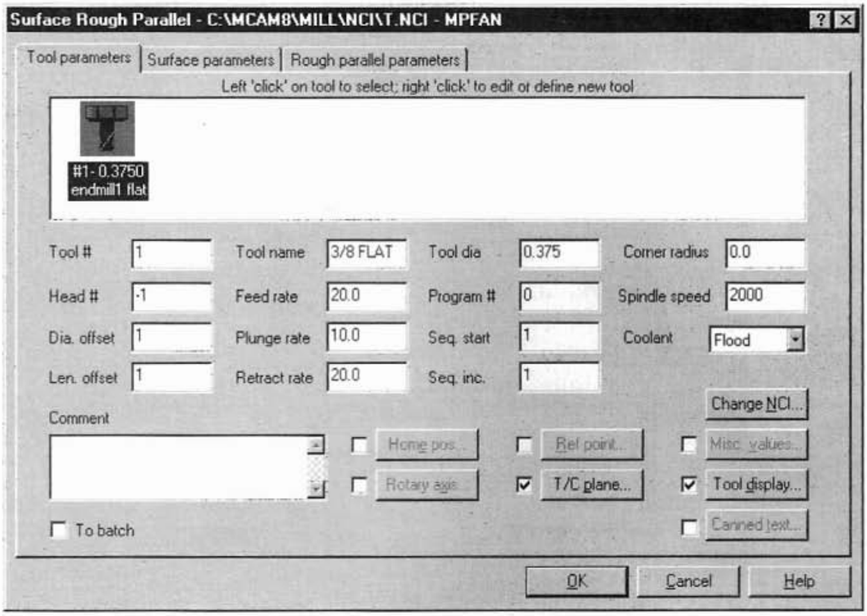
Refer to earlier chapters if you need to.

Toolpath creation

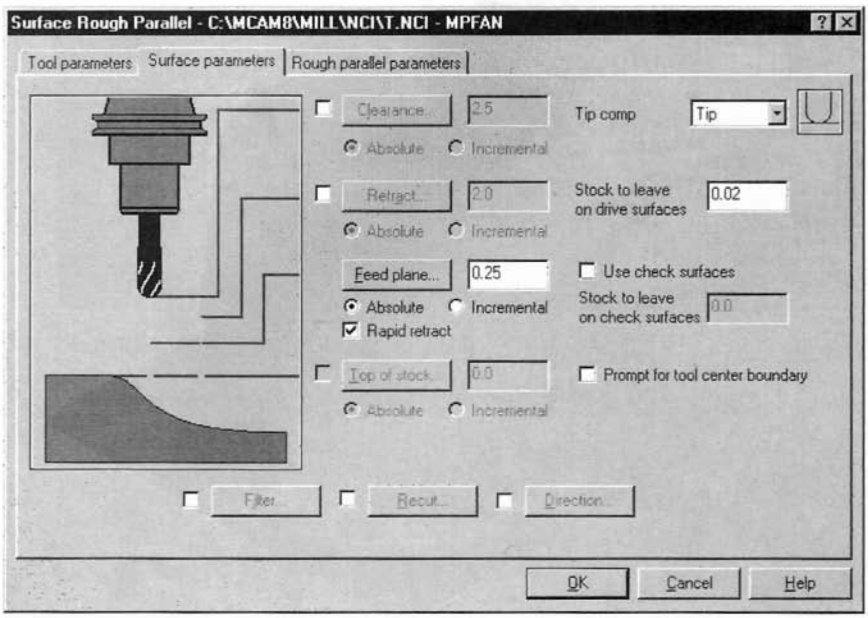
▶ *Rough out the surface*

1. Choose
   - **MAIN MENU**
   - **Toolpaths**
   - **Surface**
   - **Rough**
   - **Parallel**
   - **Cavity**
   - **All**
   - **Surfaces**
   - **Done**
2. Right-click in the tool display window, select **Get tool from library**, and choose the 3/8" flat endmill.
3. Choose **OK**.

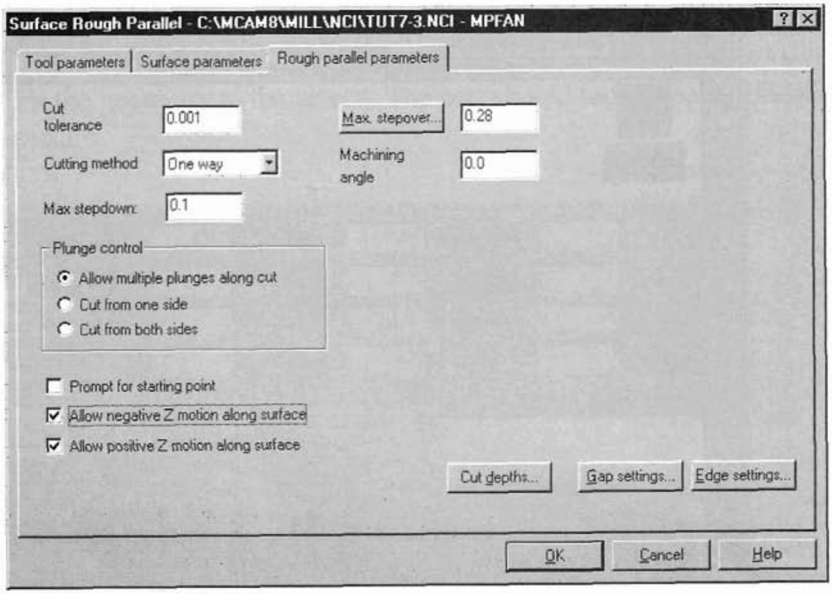
4. Enter the values shown on the following dialog box.



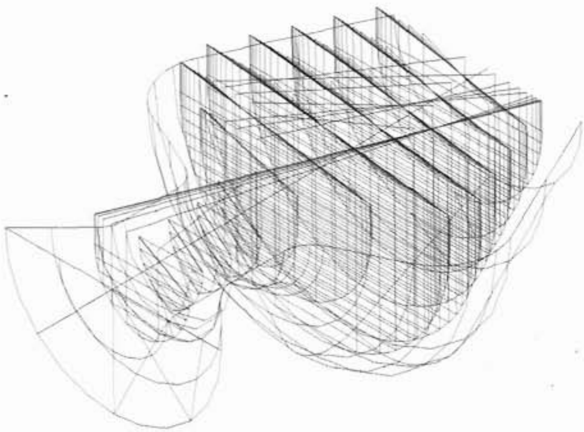
5. Select the **Surface parameters** tab and enter the values shown on the following dialog box.



6. Select the **Rough parallel parameters** tab and enter the values shown on the following dialog box.

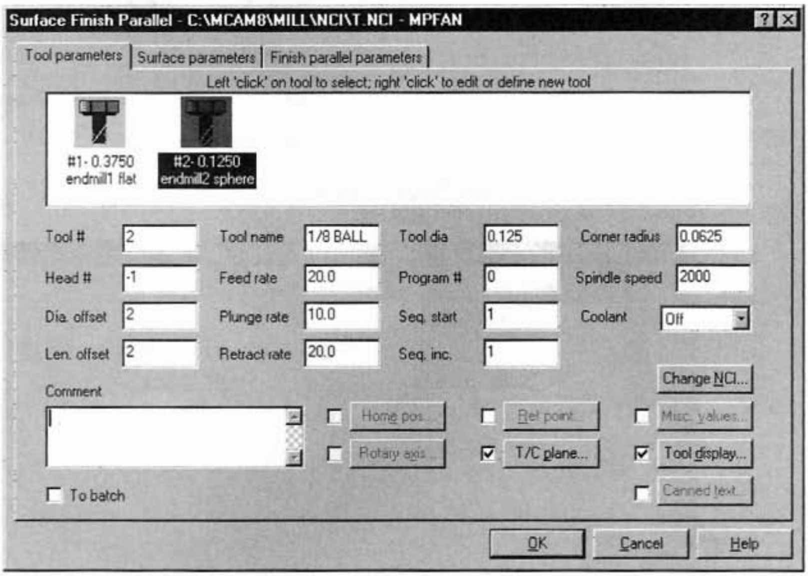


7. Select **OK**. The toolpath should look like the following picture.

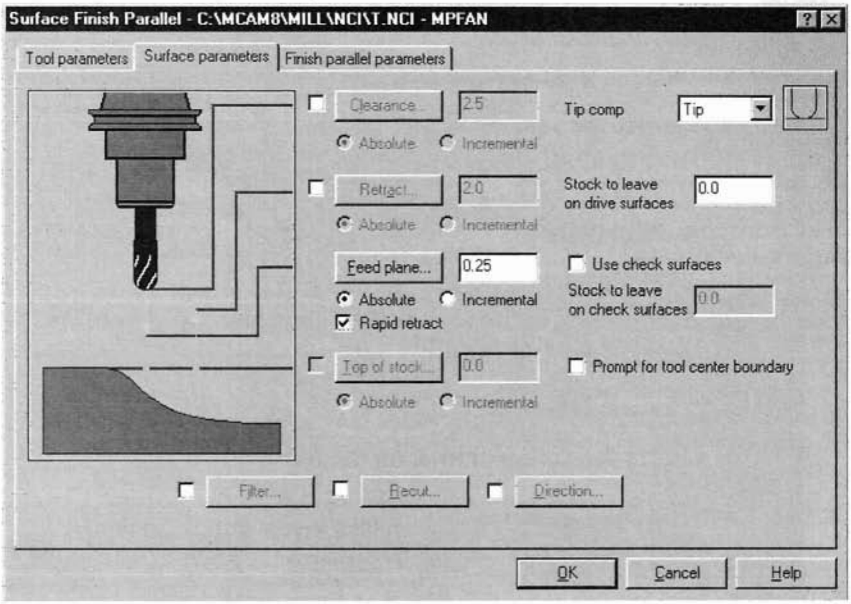


▶ *Finish the surface*

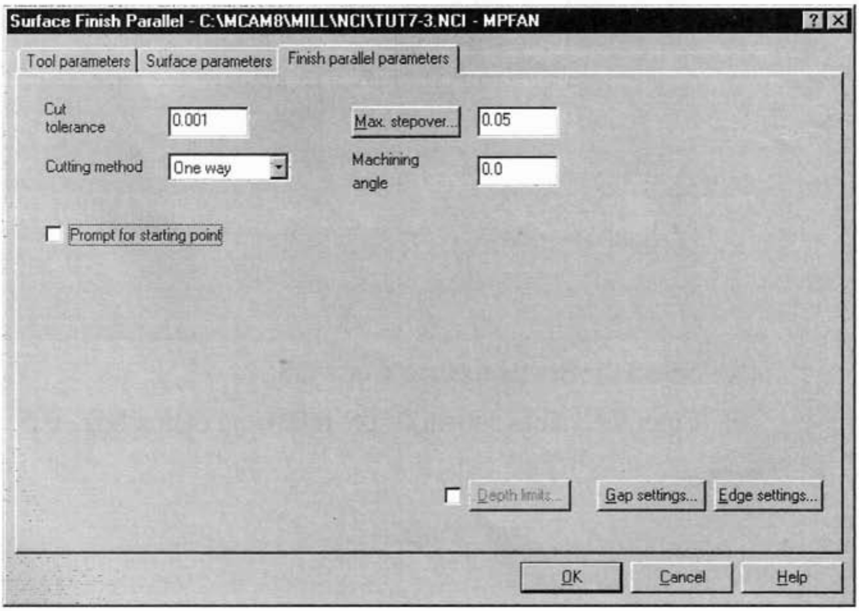
1. Choose
   - **Finish**
   - **Parallel**
   - **All**
   - **Surfaces**
   - **Done**
2. Right-click in the tool display area, select **Get tool from library** and choose the 1/8" ball endmill.
3. Choose **OK**.
4. Enter the values shown on the following dialog box.



5. Select the **Surface parameters** tab.
6. Enter the values shown on the following dialog box.



7. Select the **Finish parallel parameters** tab.
8. Enter the values shown on the following dialog box.

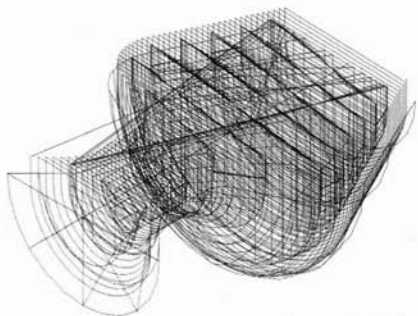


9. Select **OK**.

▶ *Backplot the toolpaths*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Toolpaths**
   - ◆ **Operations**
   - ◆ **Select All**
   - ◆ **Backplot**
   - ◆ **Run**
   - ◆ **BACKUP**

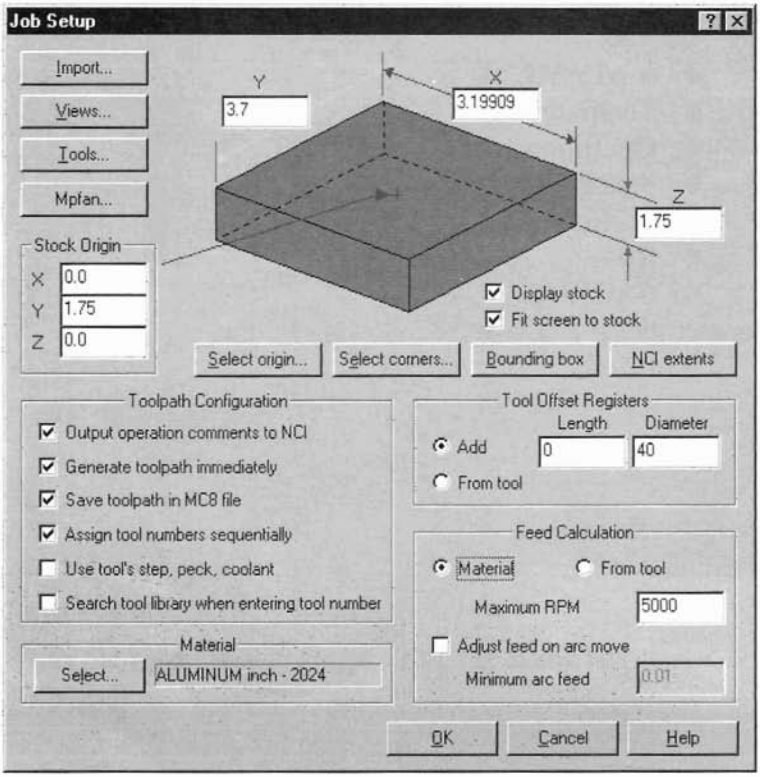
Your part should look like the following picture.



2. Choose **OK** to exit Operations Manager.

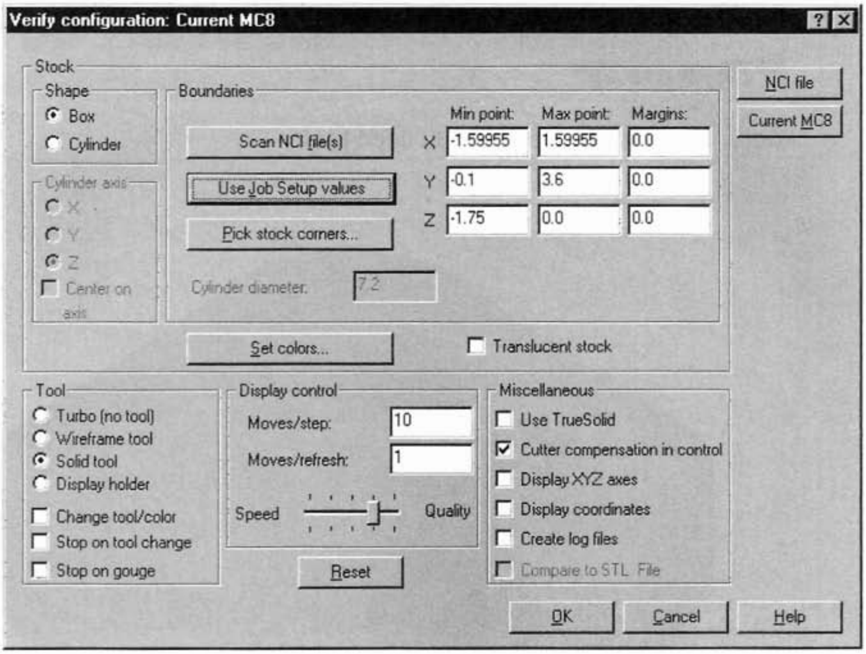
▶ *Verify the toolpath*

1. Choose **Job setup**.
2. Enter the values shown on the following dialog box.



3. Choose **OK**.
4. Choose **Operations**.
5. Choose **Verify**.

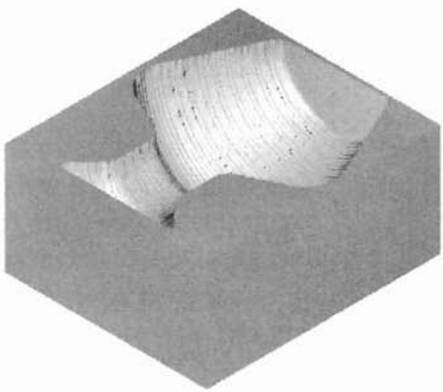
6. Choose the **Configure** button from the toolbar.
7. Choose **Use Job Setup values** button and enter the values shown on the following dialog box.



8. Choose **OK**.

9. Choose the **Machine** button from the Verify toolbar.

Your part should look like the following picture.

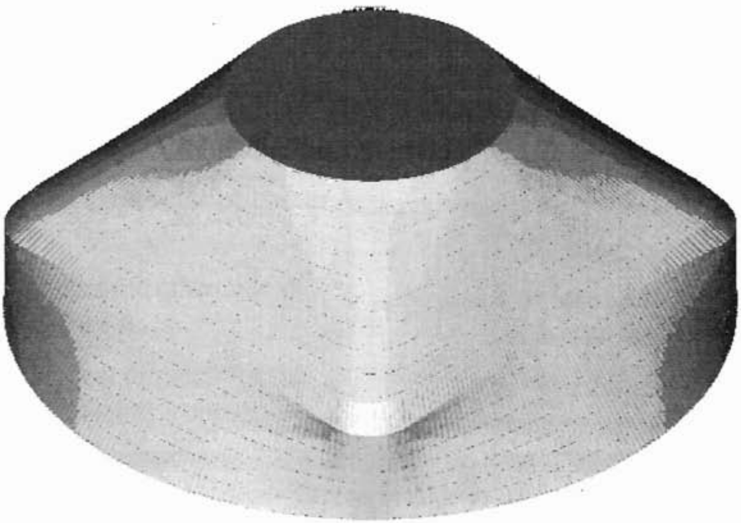


10. Exit from the Verify toolbar and the Operations Manager.

▶ *Save the file*

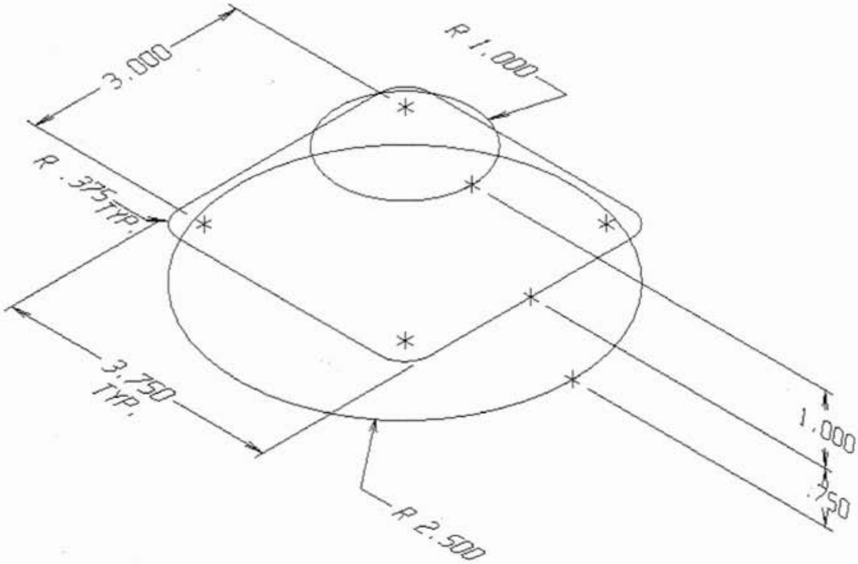
Refer to earlier chapters if you need to.

14 3D Ruled Surface - Radial



Objectives

- Design a 3D wireframe drawing.
- Create arcs using polar positioning.
- Create lines using coordinate entry.
- Create arcs tangent to 2 existing entities.
- Trim 1 entity to another entity.
- Create fillet radii.
- Create parallel lines by defining the offset direction and distance.
- Break existing geometry to create a 3D block.
- Translate existing geometry to create a 3-dimensional block.
- Create 2D pocket and contour toolpaths.
- Perform solid model verification of the toolpath.

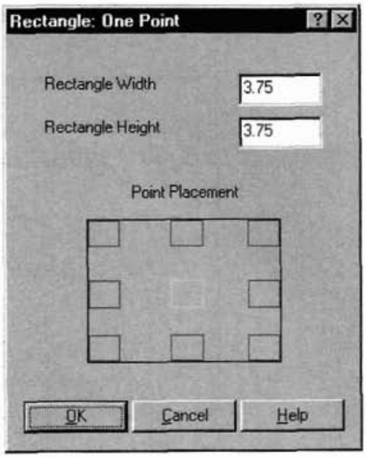


Geometry creation

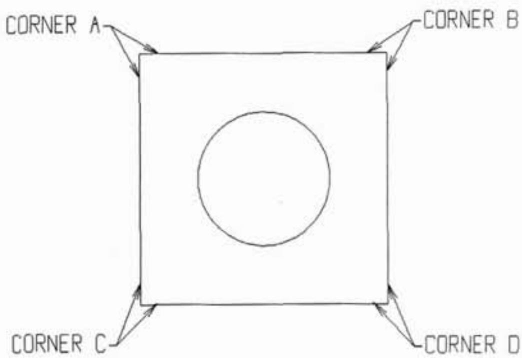
▶ *Create basic wireframe geometry*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Arc**
   - ◆ **Circ pt+rad**
2. *Enter the radius.* **1**
3. *Enter coordinates.* **0, 0**
4. Use the Secondary Menu to change the current Z depth to **–1.**
5. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Rectangle**
   - ◆ **1 point**

6. Enter the values shown in the dialog box at right.
7. Choose **OK**.
8. Choose **Origin**.
9. Choose
   - **MAIN MENU**
   - **Create**
   - **Fillet**
   - **Radius**
10. *Enter the fillet radius.* **.375**
11. Select one side of corner **A**.



12. Select the other side of corner **A.**
13. Repeat this step for the remaining 3 corners.
14. Change the current Z depth to **–1.75**.
15. Choose
   - **MAIN MENU**
   - **Create**
   - **Arc**
   - **Circ pt+rad**
16. *Enter the radius.* **2.5**
17. *Enter the coordinates.* **0, 0**

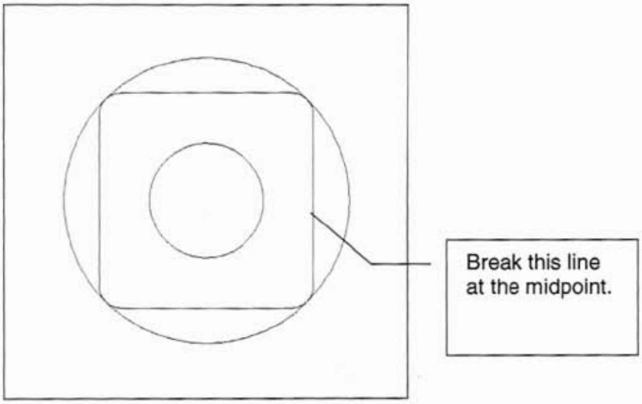


▶ *Create the surface*

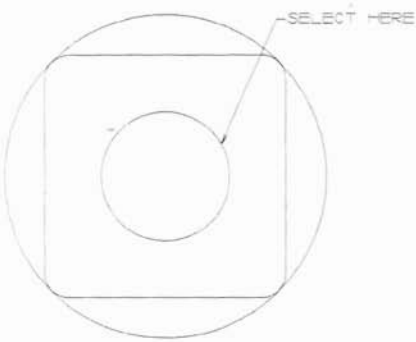
Note: To define a surface, all the start points must be lined up or the surface will become twisted and unusable.

1. Choose
   - **MAIN MENU**
   - **Modify**
   - **Break**
   - **2 pieces**

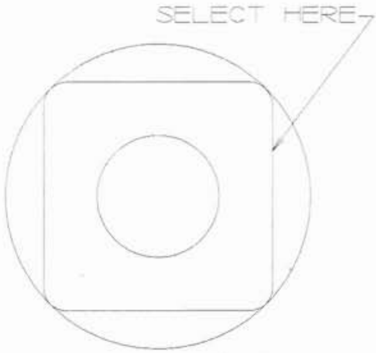
2. Select the line shown on the following picture.



3. Choose **Midpoint**.
4. Select the line again.
5. Choose
   - **MAIN MENU**
   - **Create**
   - **Surface**
   - **Ruled**
   - **Single**
6. Select the 1.0" radius (inside) just above 0 degrees as shown in the following picture.



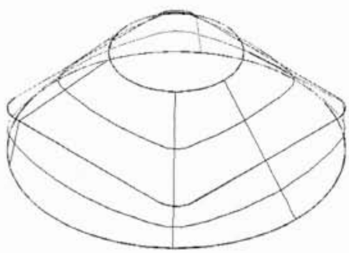
7. Choose
   - **Mode**
   - **Chain**
8. Select the broken line on the rectangle just above 0 degrees as shown in the following picture.



9. Choose
   - **Mode**
   - **Single**

Note: The chaining arrows should point up. If they don't, reverse them by choosing Reverse from the Main Menu.

10. Select the 2.5" radius (outside), just above 0 degrees.
11. Choose
    - **Done**
    - **Do it**
12. Change the Gview to isometric. Your part should look like the picture at right.



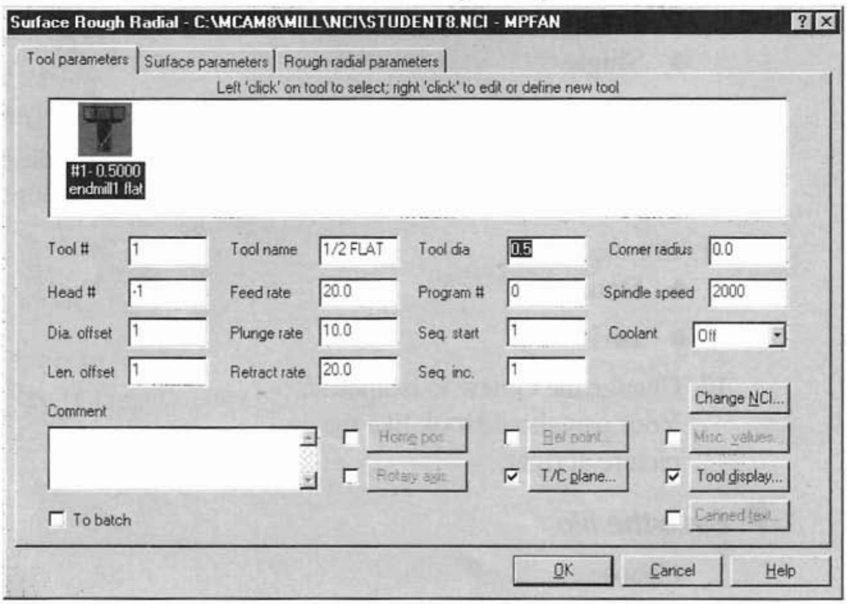
▶ *Save the file*

1. Choose
   - **MAIN MENU**
   - **File**
   - **Save**
2. Save this file using your last name followed by 14.

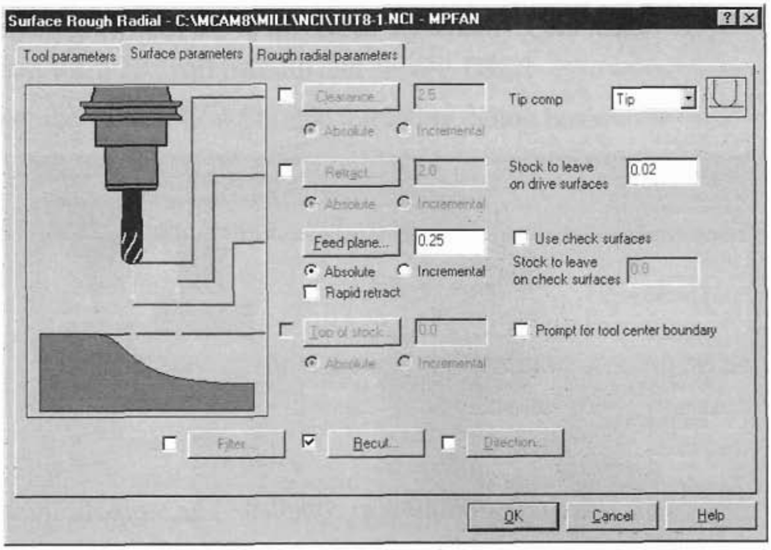
Toolpath creation

▶ *Rough out the surface*

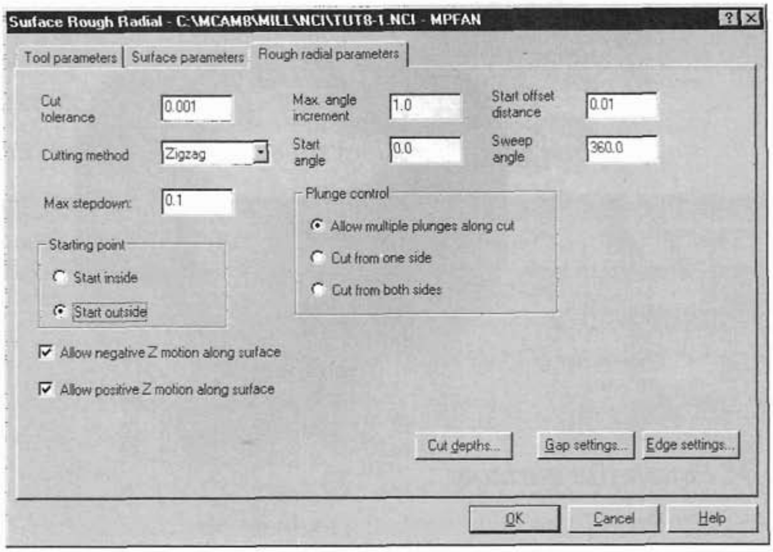
1. Choose
   - **MAIN MENU**
   - **Toolpaths**
   - **Surface**
   - **Rough**
   - **Radial**
   - **Cavity**
   - **All**
   - **Surfaces**
   - **Done**
2. Right-click in the tool display area and choose **Get tool from library**.
3. Choose the ½" flat endmill.
4. Choose **OK**.
5. Enter the values shown in the following dialog box.



6. Select the **Surface parameters** tab.
7. Enter the values shown on the following dialog box.

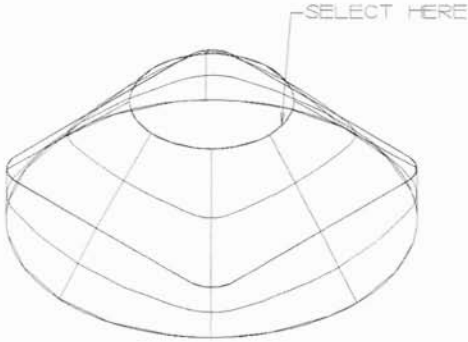


8. Select the **Rough radial parameters** tab.
9. Enter the values shown on the following dialog box.

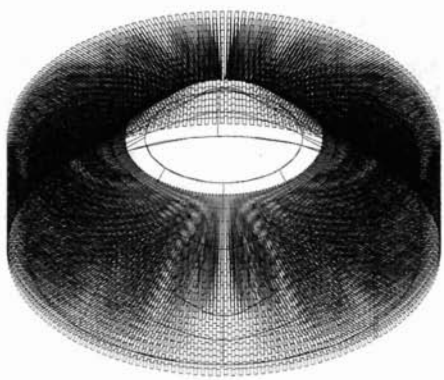


10. Choose **OK**.
11. Choose **Center**.

12. Select the 1" radius arc as shown in the following picture.



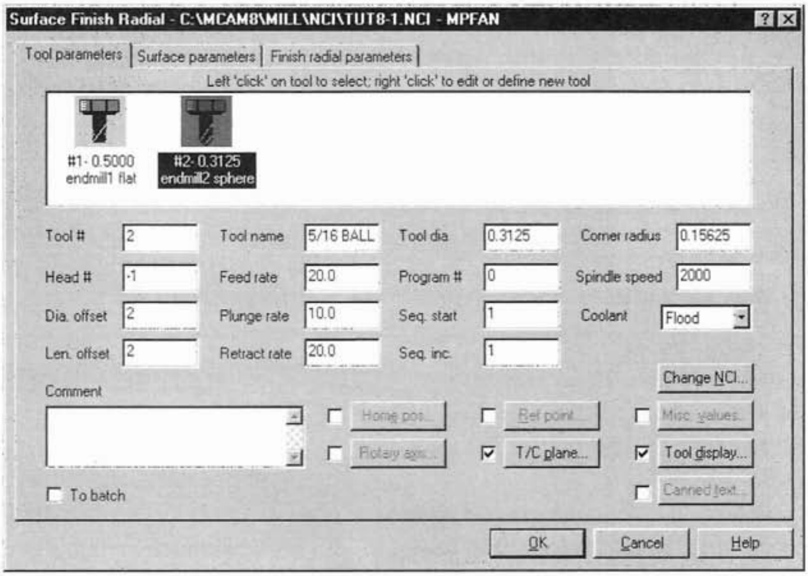
Note: This takes a few minutes to complete. The toolpath should look like the following picture.



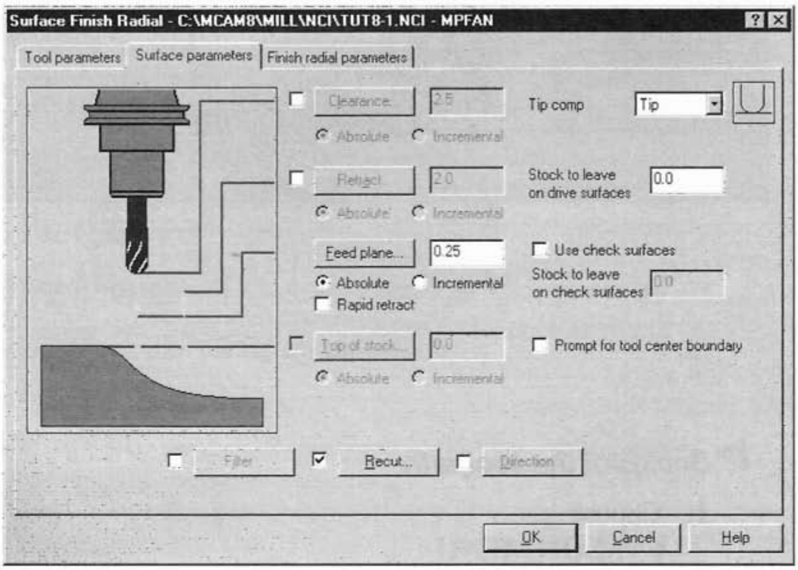
▶ *Finish the surface*

1. Press [Alt+T] to toggle the toolpath off.
2. Choose
   - ◆ **Finish**
   - ◆ **Radial**
   - ◆ **All**
   - ◆ **Surfaces**
   - ◆ **Done**

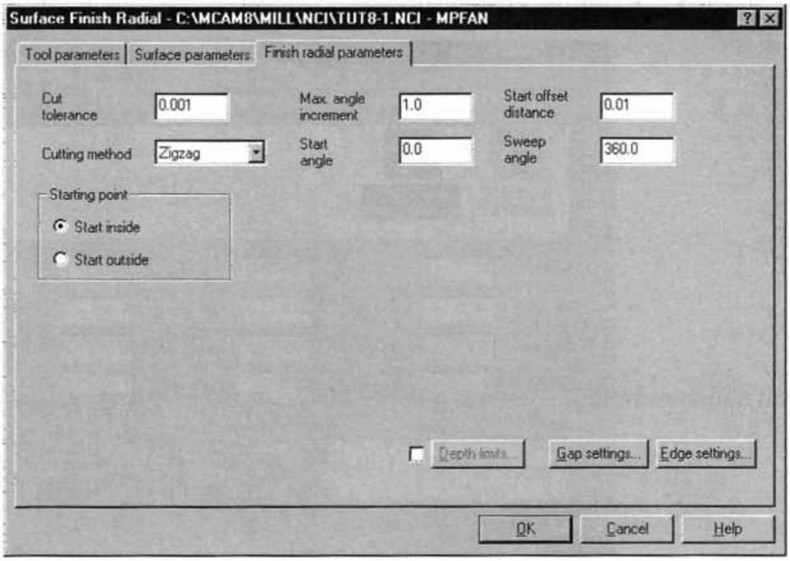
3. Right-click the tool display area and select **Get tool from library**.
4. Select the 5/16" ball endmill and choose **OK**.
5. Enter the values shown on the following dialog box.



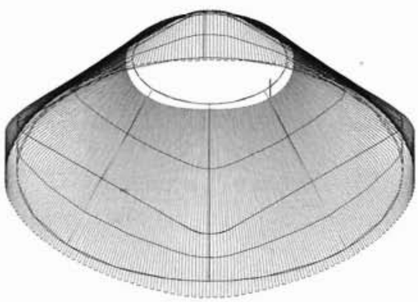
6. Select the **Surface parameters** tab.
7. Enter the values shown on the following dialog box.



8. Select the **Finish radial parameters** tab.
9. Enter the values shown on the following dialog box.



10. Choose **OK**.
11. Choose **Center**.
12. Select the 1" arc. The toolpath should look like the following picture.



▶ *Backplot the toolpath*

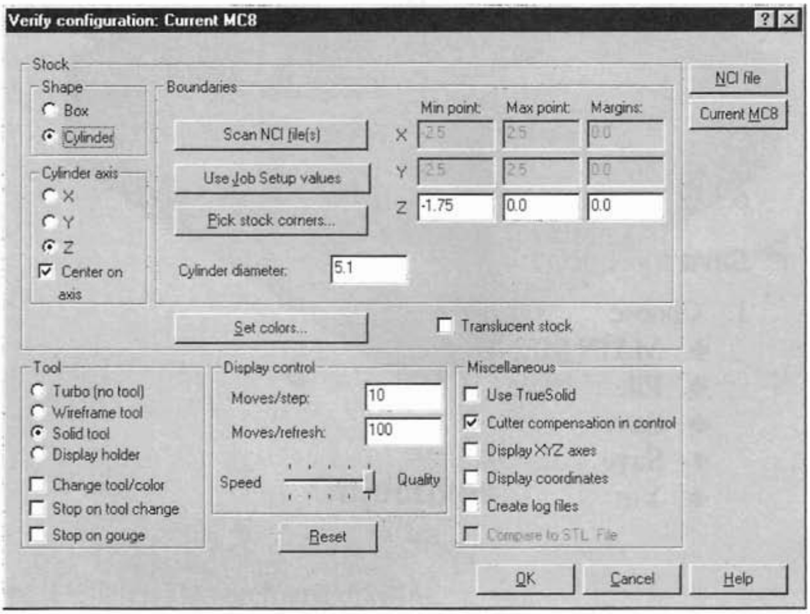
1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Toolpaths**

- **Operations**
- **Select All**
- **Backplot**
- **Run**

2. Choose **BACKUP** to return to the Operations Manager.

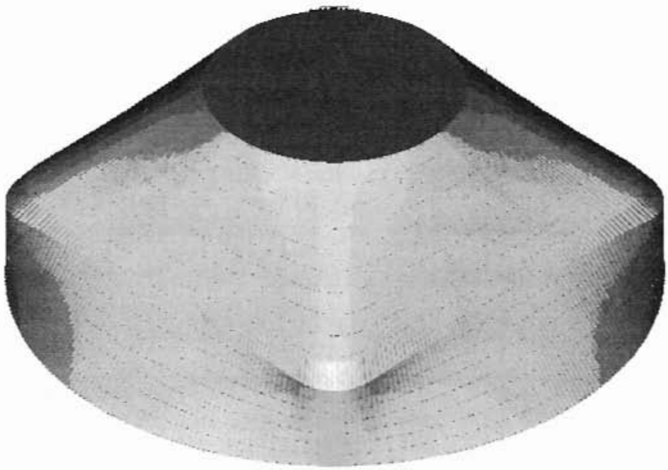
▶ *Verify the toolpath*

1. Choose **Verify**.
2. Choose **Configure**.
3. Enter the values shown on the following dialog box.



4. Choose **OK**.
5. Choose **Machine**.

The part should look like the following picture.

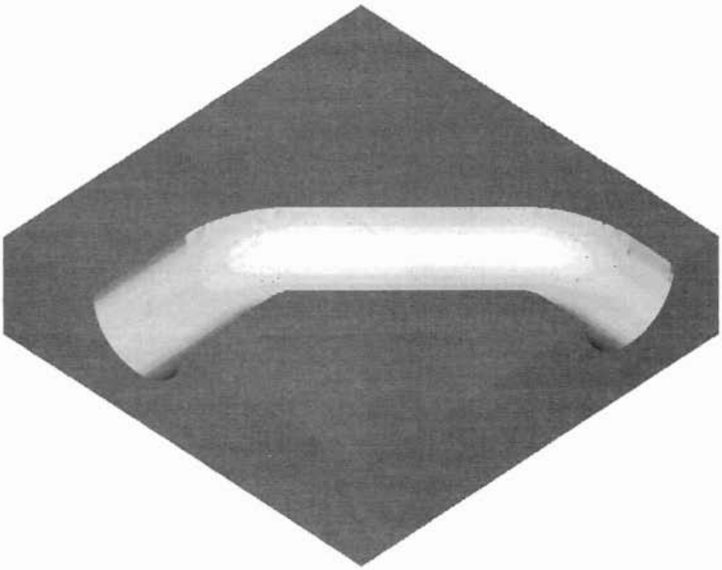


6. Choose **OK** to exit the Operations Manager.

▶ *Save the updated file*

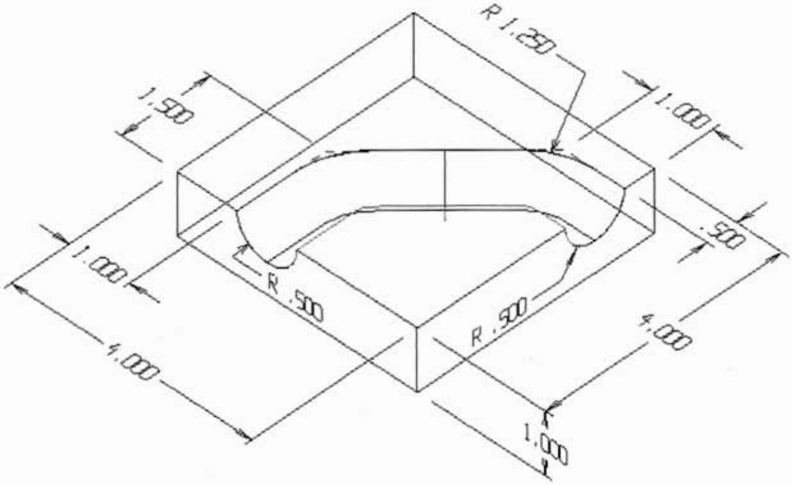
1. Choose
   - ◆ **MAIN MENU**
   - ◆ **File**
   - ◆ **Save**
   - ◆ **Save**
   - ◆ **Yes**

15 3D Swept Surface - Flowline



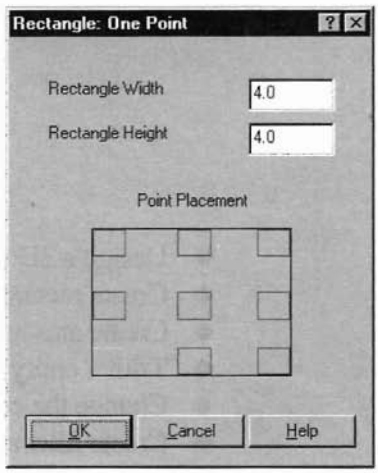
Objectives

- Design a 3D wireframe drawing.
- Create rectangles using the coordinate positioning.
- Create arcs and lines using polar positioning.
- Trim 1 entity to two existing entities.
- Change the construction plane of the drawing entities.
- Create fillet radii.
- Create a swept surface.
- Change the view of the part.
- Create a 3D toolpath.
- Perform solid model verification of the toolpath.



Geometry creation

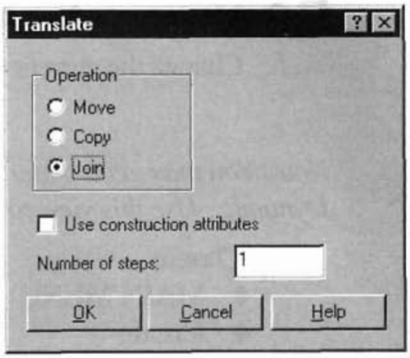
▶ *Create the block*



1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Rectangle**
   - ◆ **1 point**
2. Enter the values shown in the dialog box at right, including the lower-left Point Placement.
3. Choose **OK**.
4. Choose **Origin**.
5. Choose the **Fit** button from the toolbar.
6. Choose
   - ◆ **MAIN MENU**
   - ◆ **Xform**

- ◆ **Translate**
- ◆ **All**
- ◆ **Entities**
- ◆ **Done**
- ◆ **Rectang**

7. *Enter the translation vector.* **Z-1**
8. Enter the values as shown.
9. Choose **OK**.
10. Change the Gview to **Isometric**.
11. Fit the geometry to the screen.



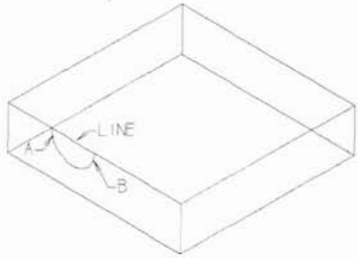
▶ *Create the 0.5 radius arc*

1. Change the Cplane to **Front**.
2. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Arc**
   - ◆ **Polar**
   - ◆ **Center pt**
3. *Enter the coordinates.* **1.5, 0**
4. *Enter the radius.* **0.5**
5. *Enter the initial angle.* **180**
6. *Enter the final angle.* **0**

▶ *Trim the line inside the arc*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Modify**
   - ◆ **Trim**
   - ◆ **Divide**

2. Select the line inside the 0.5 radius arc.
3. Select the end of the arc at **A**.
4. Select the end of the arc at **B**.

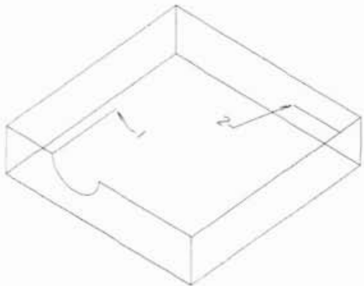


▶ *Create the contour profile*

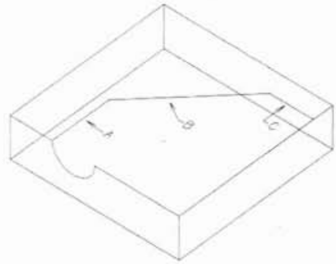
1. Change the current Cplane to **Top**.

Note: You may get a better view of the geometry by changing the Gview to Dynamic. Use this view to manually rotate the image.

2. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Line**
   - ◆ **Polar**
3. *Specify an endpoint.* **1, 0**
4. *Enter the angle in degrees.* **90**
5. *Enter the line length.* **1.5**
6. *Specify an endpoint.* **4, 4-0.5**
7. *Enter the angle in degrees.* **180**
8. *Enter the line length.* **1.0**
9. Choose
   - ◆ **BACKUP**
   - ◆ **Endpoints**
10. Select the endpoint of the polar line at **1** as shown at right.
11. Select the endpoint of the other polar line at **2**.
12. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Fillet**
   - ◆ **Radius**

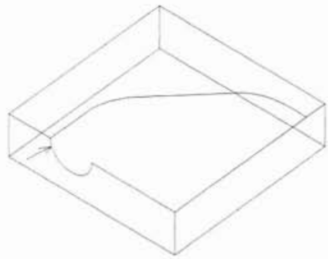


13. *Enter the fillet radius.* **1.25**
14. Select line **A**.
15. Select line **B**.
16. Select line **B**.
17. Select line **C**.

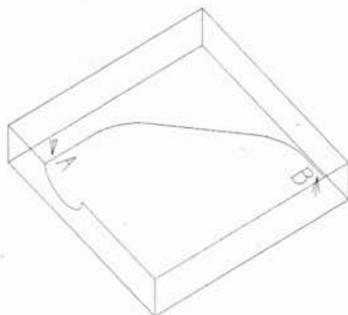


▶ *Create the swept surface*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Surface**
   - ◆ **Sweep**
   - ◆ **Single**



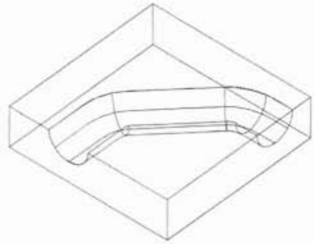
2. Select the arc on the left side.
3. Choose
   - ◆ **Done**
   - ◆ **Chain**
   - ◆ **Partial**
4. Select Entity **A** near the arc.
5. Select Entity **B** near the edge of the rectangle.



6. Choose
   - ◆ **Done**
   - ◆ **Do it**

▶ *Save the file*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **File**
   - ◆ **Save**

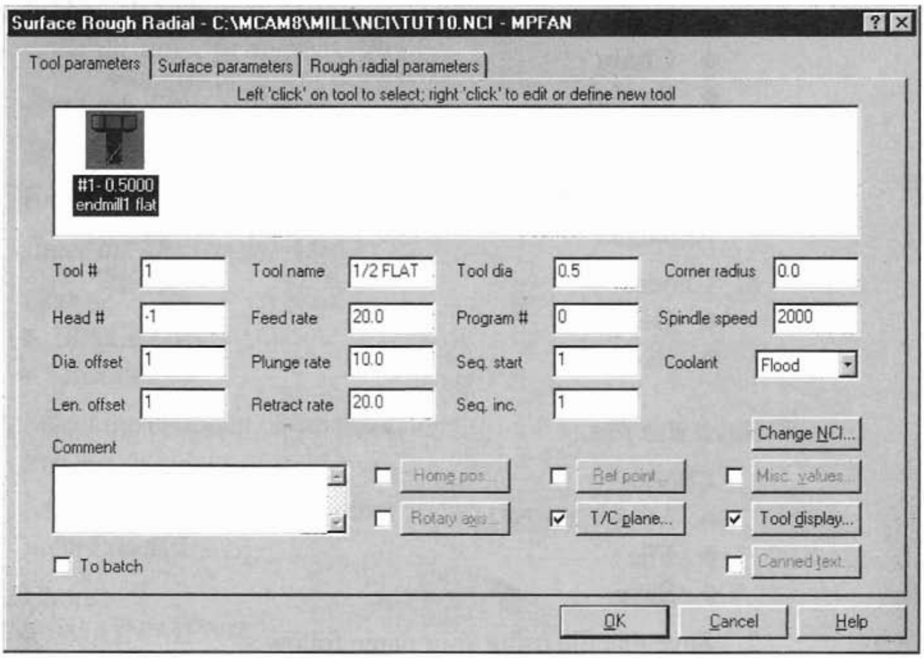


2. Save this file using your name followed by 15.

Toolpath creation

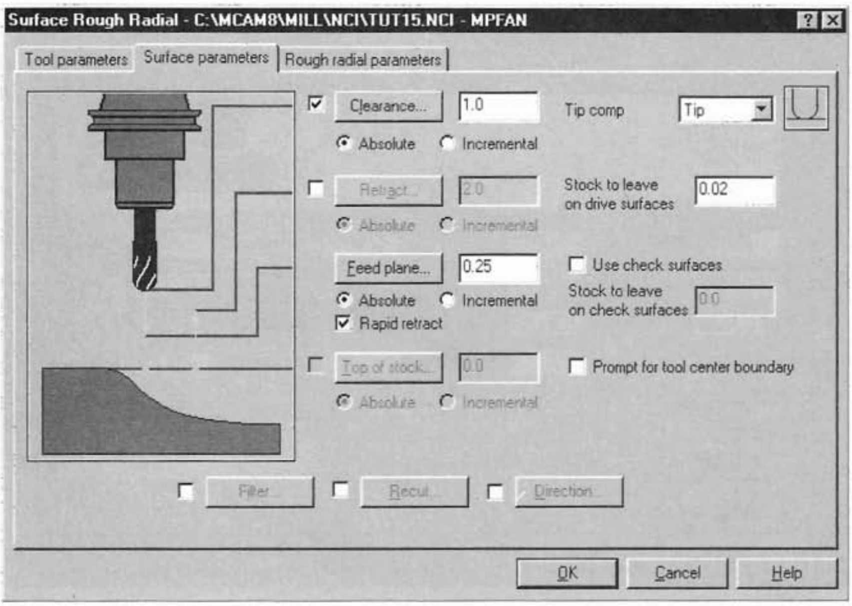
▶ *Rough out the surface*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Toolpaths**
   - ◆ **Surface**
   - ◆ **Rough**
   - ◆ **Radial**
   - ◆ **Cavity**
   - ◆ **All**
   - ◆ **Surfaces**
   - ◆ **Done**
2. Right-click in the tool display area and select **Get tool from library**.
3. Select the ½" flat endmill and choose **OK**.
4. Enter the values shown on the following dialog box.



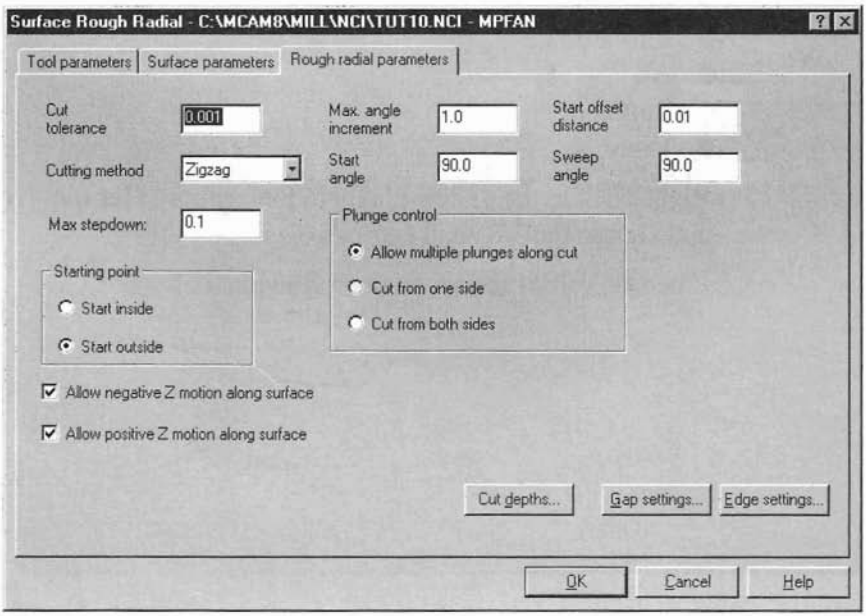
5. Select the **Surface parameters** tab.

6. Enter the values shown on the following dialog box.



7. Select the **Rough radial parameters** tab.

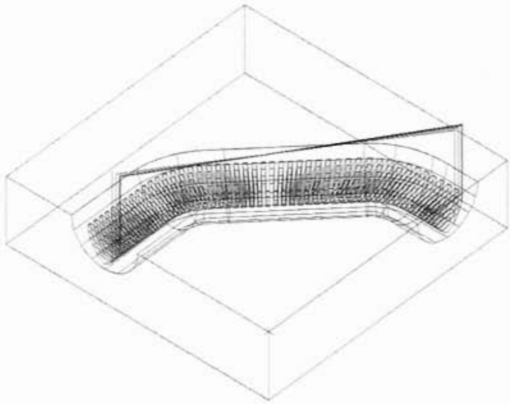
8. Enter the values shown on the following dialog box.



9. Choose **OK**.

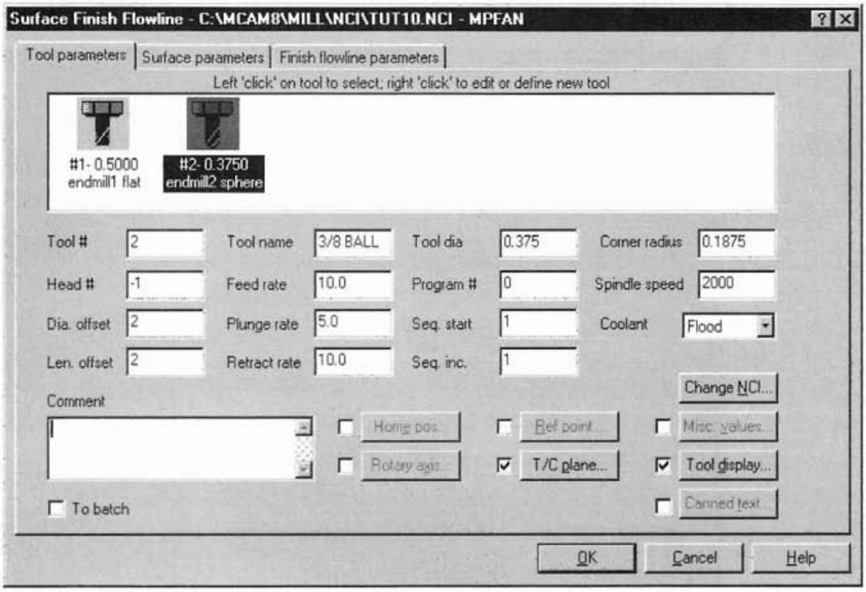
10. *Enter the rotation point.* **4, 0, 0**

The toolpath should look like the following picture.

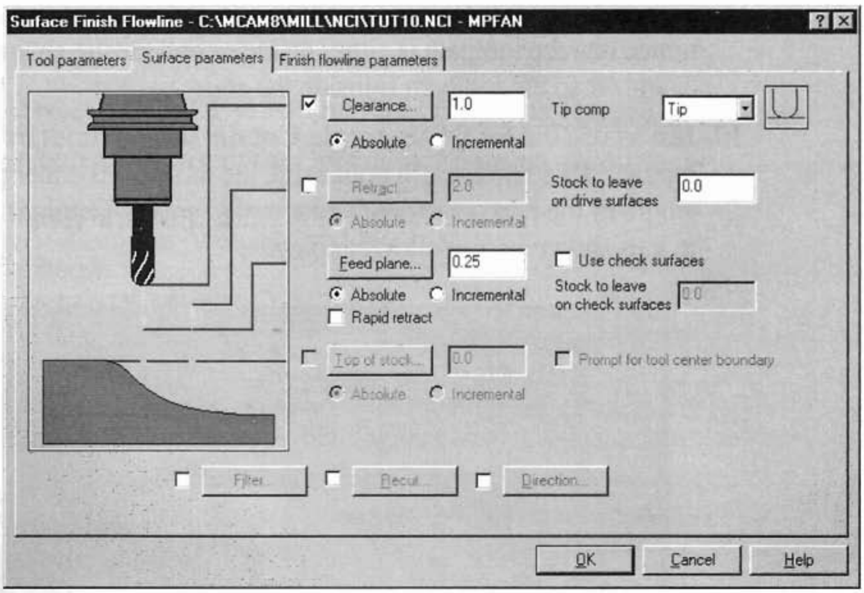


Finish the surface

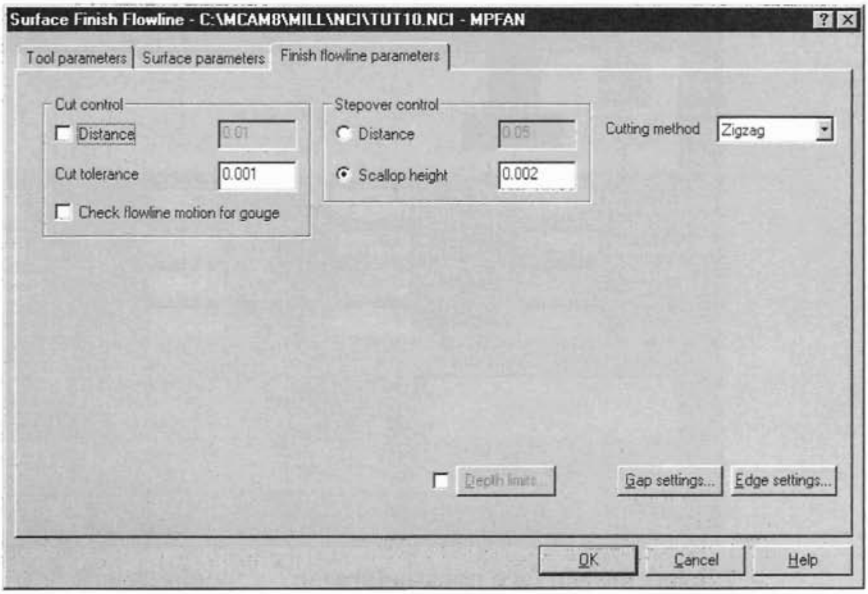
1. Choose
   - **Finish**
   - **Flowline**
   - **All**
   - **Surfaces**
   - **Done**
2. Right-click in the tool display window, choose **Get tool from library**, and choose the 3/8" ball endmill.
3. Enter the values shown on the following dialog box.



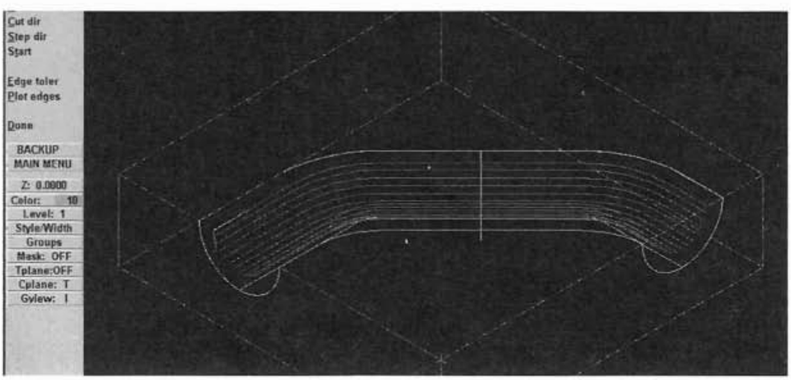
4. Select the **Surface parameters** tab.
5. Enter the values shown on the following dialog box.



6. Select the **Finish flowline pararmeter** tab.
7. Enter the values shown on the following dialog box.

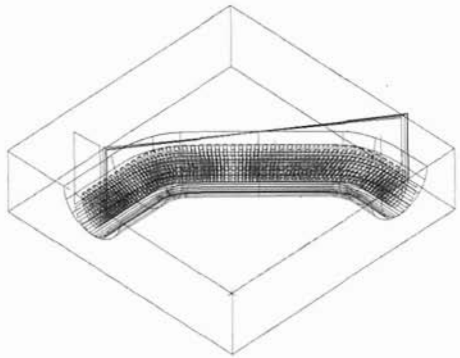


8. Choose **OK**.
9. Toggle **Offset** by clicking it with your mouse several times. You'll notice that the toolpath is either inside or outside the chained area. Toggle it so the toolpath is inside the area.
10. Just as you did for Offset, toggle **Cut dir** several times by clicking your mouse. This time, you'll notice that the toolpath is either parallel to the length of the part or perpendicular to the length. Toggle it so it is parallel to it as shown in the following picture.



11. Choose **Done**.

12. The completed toolpath should look like the following picture.



Backplot the toolpath

1. Press [Alt+O] to open the Operations Manager.

2. Choose
   - **Select All**
   - **Backplot**
   - **Run**

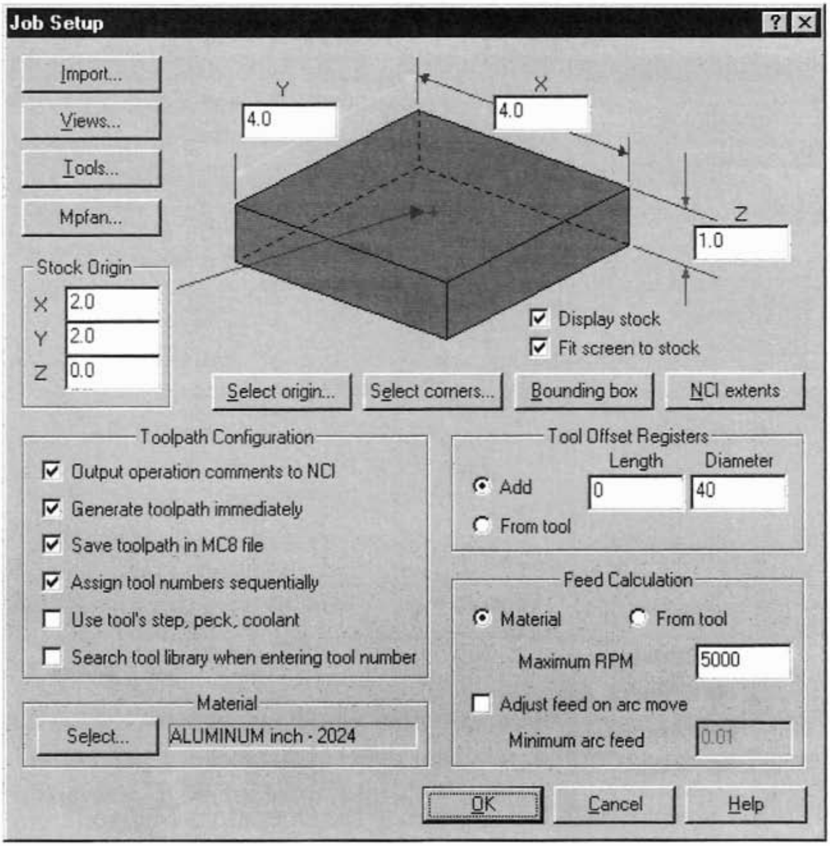
3. Select **BACKUP** to return to the Operations Manger.

4. Choose **OK** to exit the Operations Manager.

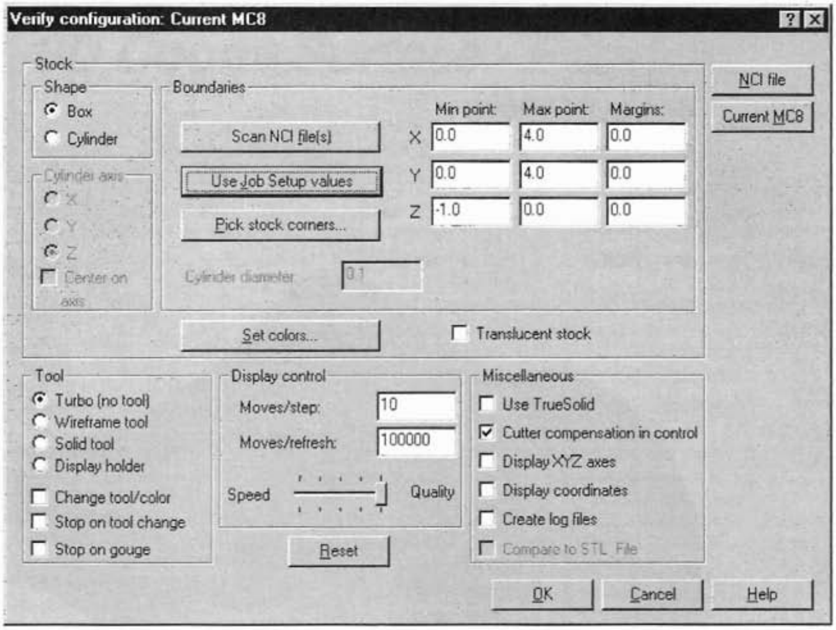
Verify the toolpath

1. Choose
   - **MAIN MENU**
   - **Toolpaths**
   - **Job setup**

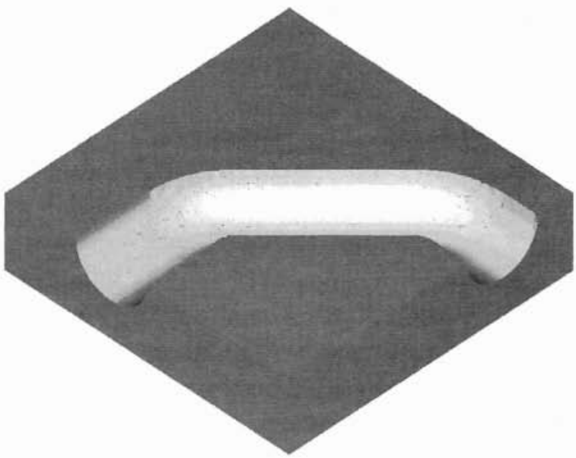
2. Enter the values shown on the following dialog box.



3. Choose **OK**.
4. Choose **Operations**.
5. Choose **Verify**.
6. Choose the **Configure** button.
7. Choose the **Use Job Setup values** button and enter the values shown on the following dialog box.



8. Choose the **Machine** button. The verified part should look like the following picture.



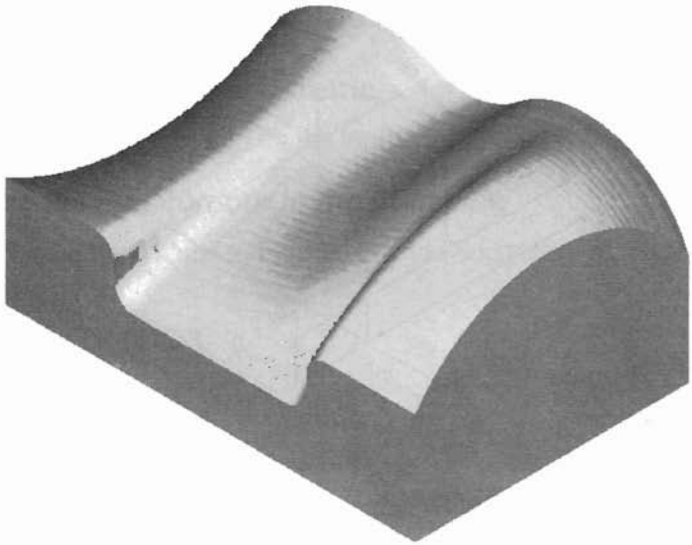
9. Exit the Verify toolbar.
10. Choose **OK** to exit the Operations Manager.

▶ *Save the updated file*

1. Choose
   - **MAIN MENU**
   - **File**
   - **Save**
   - **Save**
   - **Yes**

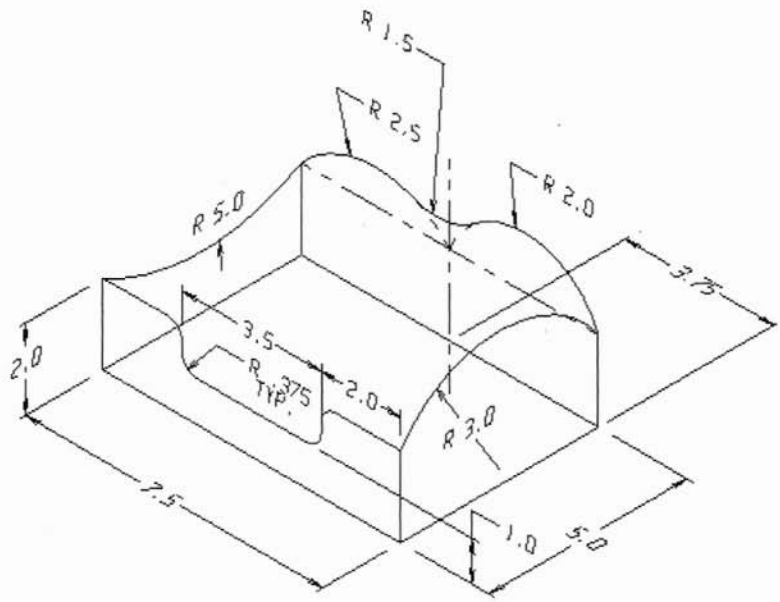
16 3D Coons Surface - Flowline

Por Steven Anson Coons.



Objectives

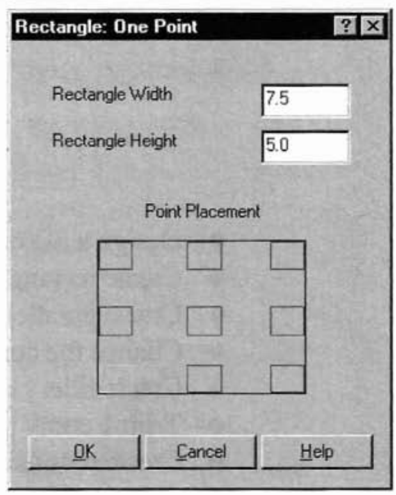
- Design a 3D drawing.
- Create rectangles using coordinate positioning.
- Create parallel lines by defining the offset direction and distance.
- Change the construction depth and plane of the drawing.
- Create fillet radii.
- Trim 1 entity to 2 existing entities.
- Create arcs using coordinate positioning.
- Create a Coons surface.
- Create a 3D milling toolpath.
- Perform solid model verification of the toolpath.



Geometry creation

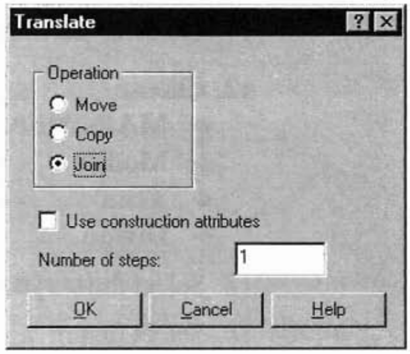
▶ *Create the block*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Rectangle**
   - ◆ **1 point**
2. Enter the parameters shown at right.
3. Choose **OK**.
4. Choose **Origin**.
5. Choose the **Fit** button from the toolbar.
6. Choose
   - ◆ **MAIN MENU**
   - ◆ **Xform**



- **Translate**
- **All**
- **Entities**
- **Done**
- **Rectang**

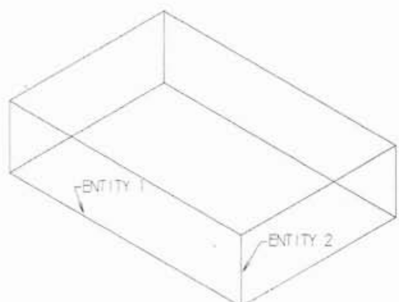
7. *Enter the translation vector.* **Z2.0**
8. Enter the values shown at right.



9. Choose **OK**.
10. Change the Gview to **Isometric**.
11. Choose the **Fit** button on the toolbar.

▶ *Create the first contour in the front view*

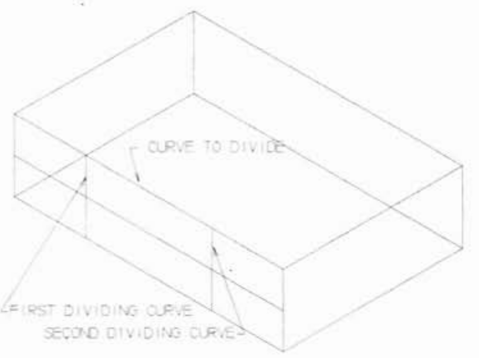
1. Change the Cplane to **Front** using the Secondary Menu.
2. Choose
   - **MAIN MENU**
   - **Create**
   - **Line**
   - **Parallel**
   - **Side/dist**
3. Select entity **1**.
4. Select a point well above entity 1.
5. Enter the parallel line distance. **1**



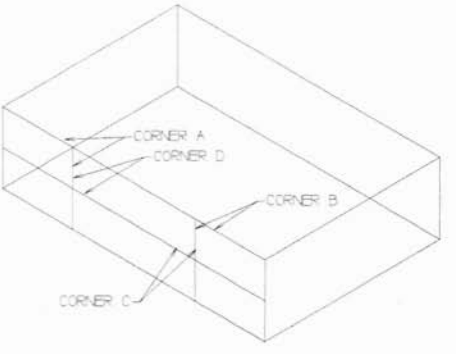
Note: If the created line appears below Entity 1, it means you selected a point too close to Entity 1. Delete the line and try again, choosing a point well above Entity 1.

6. Select entity **2**.
7. Select left of entity 2.
8. *Enter the parallel line distance.* **2**
9. Select the entity just created.
10. Select to the left of it.

11. *Enter the parallel line distance.* **3.50**
12. Choose
    - **MAIN MENU**
    - **Modify**
    - **Trim**
    - **Divide**

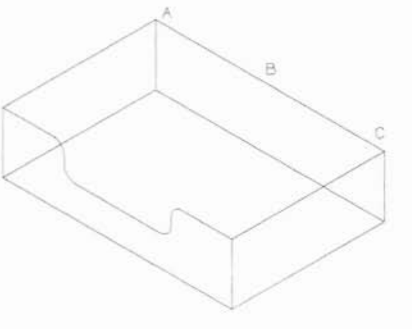


13. Select the curve to divide.
14. Select first dividing curve.
15. Select second dividing curve.
16. Choose
    - **MAIN MENU**
    - **Create**
    - **Fillet**
    - **Radius**
17. *Enter the fillet radius.* **.375**
18. Select one side of corner **A**.
19. Select the other side of corner **A.**
20. Repeat this step for the remaining 3 corners.

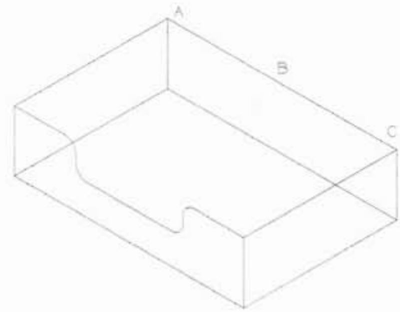


▶ *Create the second contour in the front view*

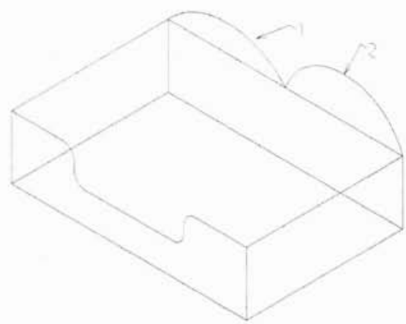
1. Select Z depth from the Secondary Menu and enter **–5.**
2. Choose
    - **MAIN MENU**
    - **Create**
    - **Arc**
    - **Endpoints**
3. Move your mouse to point A, clicking when the AutoCursor locks on Endpoint.



4. Move your mouse along the line toward B and C until the AutoCursor highlights Midpoint. Click on Midpoint.
5. *Enter the radius.* **2.5**
6. Select arc **1**.
7. Move your mouse to C, clicking when the AutoCursor locks on Endpoint.

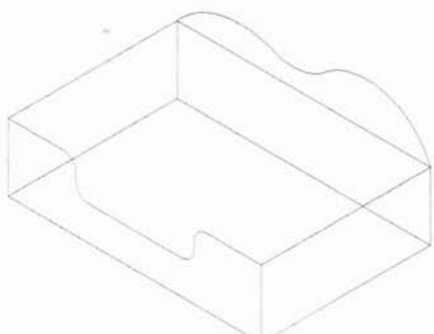


8. Move the mouse to B, clicking when the AutoCursor locks on Endpoint.
9. *Enter the radius.* **2.**
10. Select arc **2**.
11. Choose
    - **MAIN MENU**
    - **Create**
    - **Fillet**
    - **Radius**



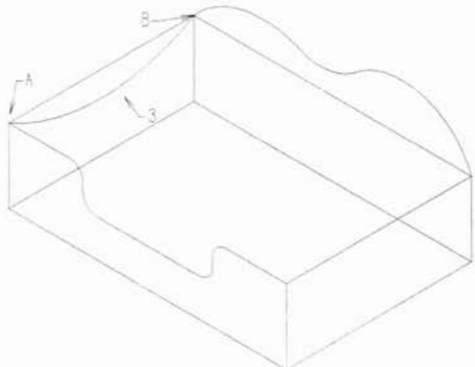
12. *Enter the fillet radius.* **1.5**
13. Select the **2.5"** radius arc near the previous point B intersection.
14. Select the **2.0"** radius arc near the previous point B intersection.

Your part should look like the following picture.



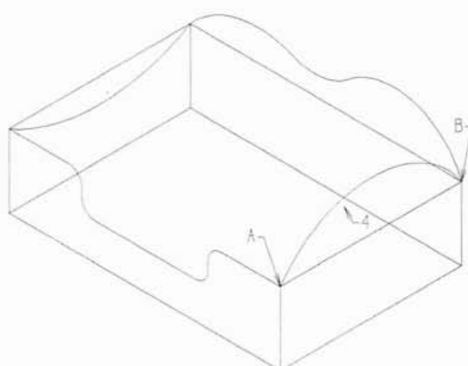
▶ Create the first contour in the side view

1. Change the current Cplane to **Side**.
2. Change the current **Z** depth to **0.0**
3. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Arc**
   - ◆ **Endpoints**
4. Using your mouse, select point **A**, clicking when the AutoCursor locks on Endpoint.
5. Select point **B**, clicking when the AutoCursor locks on Endpoint.
6. *Enter the radius.* **5.0**
7. Select arc **3**.



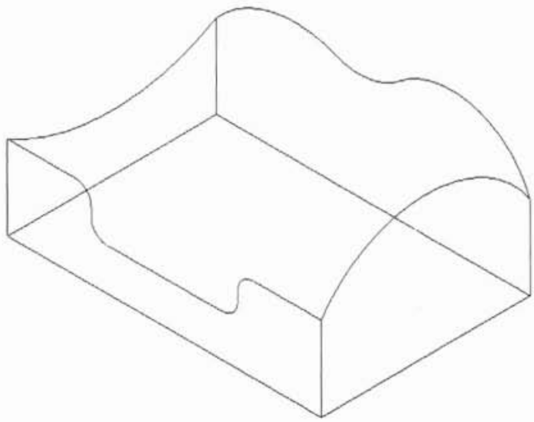
▶ Create the second contour in the side view.

1. Change the Z depth to **7.5**.
2. Select point **A**.
3. Select point **B**.
4. *Enter the radius.* **3**
5. Select arc **4**.



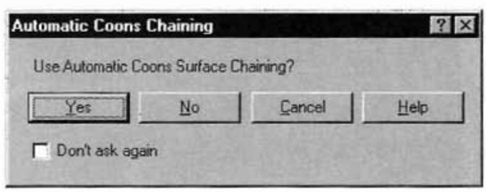
▶ *Delete the construction lines*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Delete**
2. Select the 3 construction lines. The part should look like the following picture.

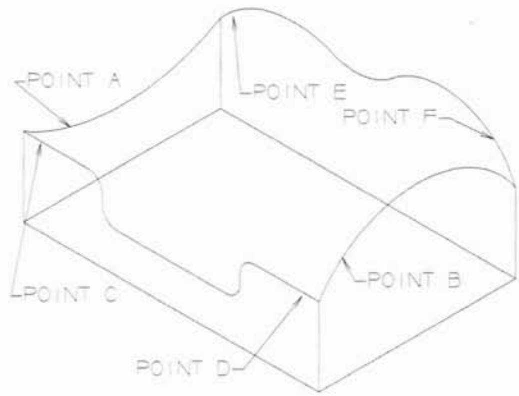


▶ *Create the Coons surface*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Surface**
   - ◆ **Coons**
2. Select **No** in the dialog box shown at right.



3. *Enter the number of patches in the along direction.* **1**
4. *Enter the number of patches in the across direction.* **1**
5. Choose **Single**.



6. Select point **A**.
7. Select point **B**.
8. Choose
   - **Mode**
   - **Chain**
   - **Partial**
9. Select point **C**.
10. Select point **D**.
11. Choose **Partial**
12. Select point **E**.
13. Select point **F**.
14. Choose
    - **Done**
    - **Do it**

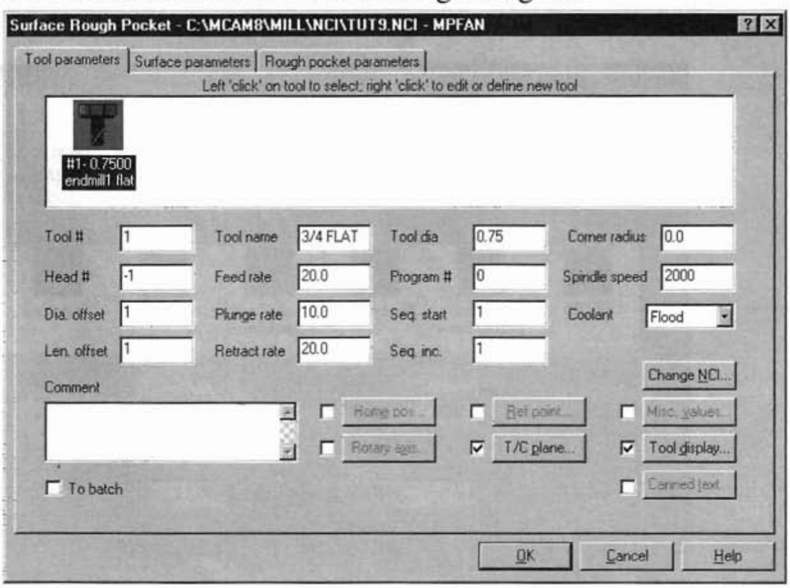
▶ *Save the file*

1. Choose
   - **MAIN MENU**
   - **File**
   - **Save**
2. Save this file using your last name followed by 16.

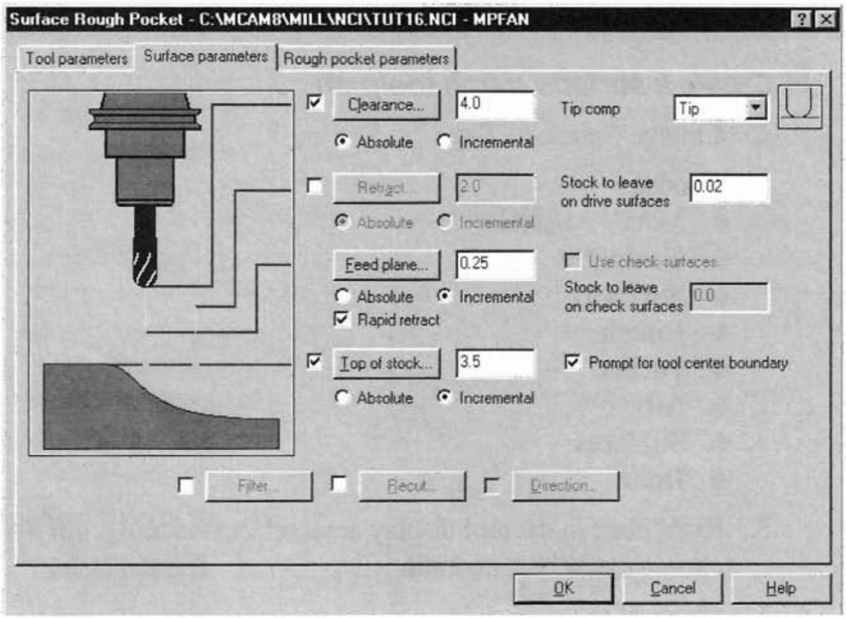
Toolpath creation

▶ *Create a surface rough toolpath*

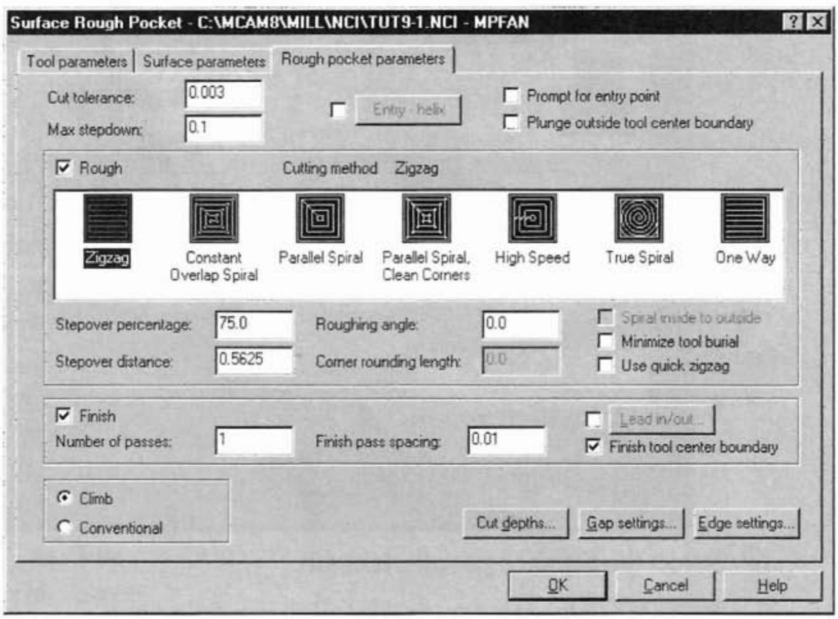
1. Change Cplane to **Top**.
2. Choose
   - **MAIN MENU**
   - **Toolpaths**
   - **Surface**
   - **Rough**
   - **Pocket**
   - **All**
   - **Surfaces**
   - **Done**
3. Right click in the tool display area, select **Get tool from library** and choose the ¾" flat endmill.
4. Choose **OK**.
5. Enter the values shown in the following dialog box.



6. Select the **Surface parameters** tab.
7. Enter the values shown on the following dialog box.

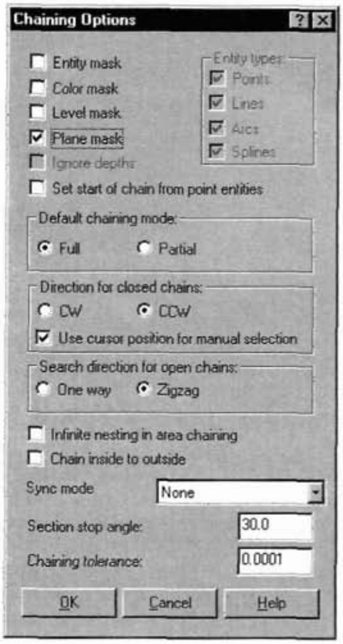


8. Select the **Rough pocket parameters** tab.
9. Enter the values shown on the following dialog box.

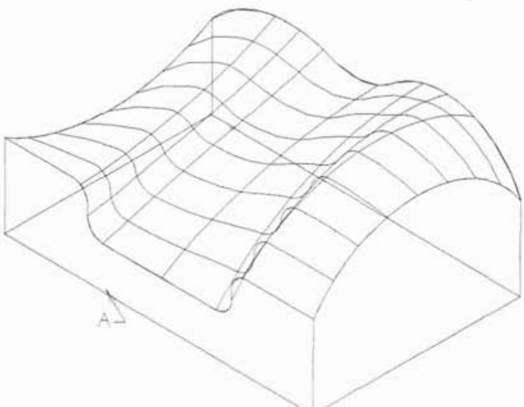


10. Choose **OK**.

11. Choose
    - ◆ **Chain**
    - ◆ **Options**
12. Select **Plane mask** in the following dialog box.

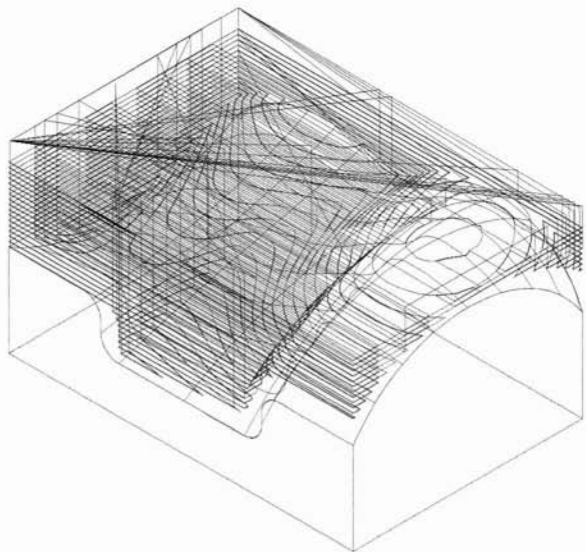


13. Choose **OK**.
14. Select entity **A** as shown in the following picture.



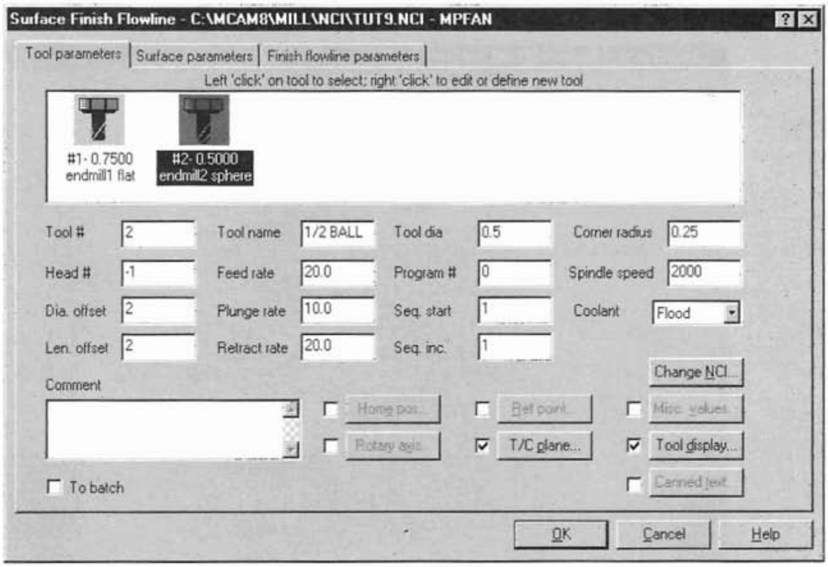
15. Choose **Done**.

The toolpath should look like the following picture.

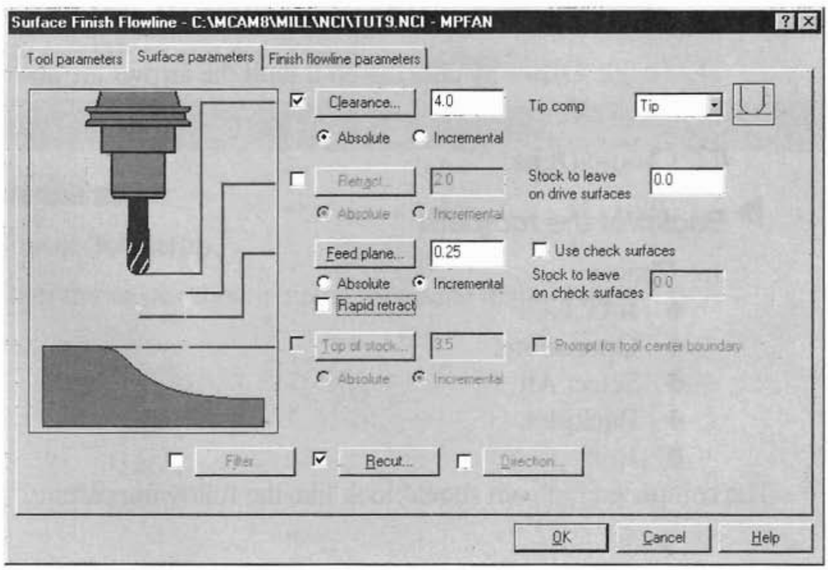


▶ *Finish the surface*

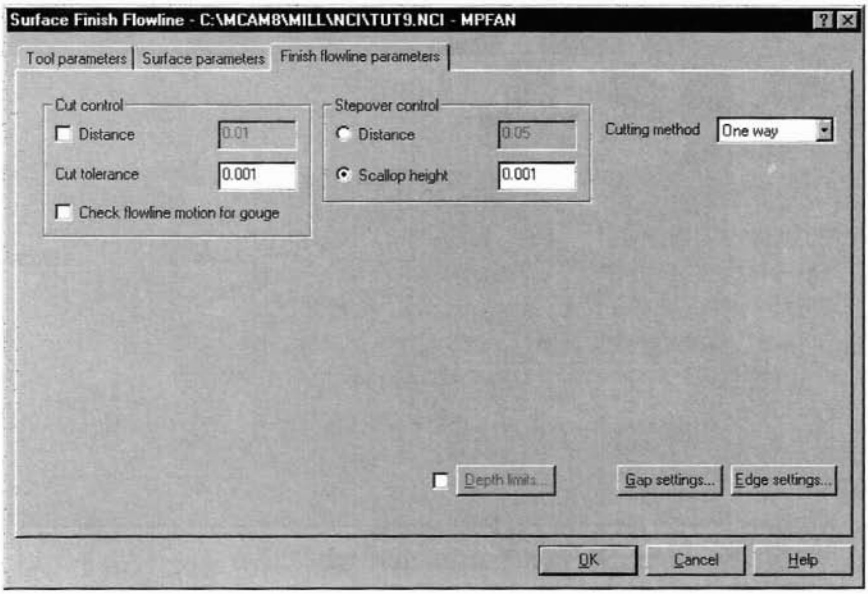
1. Press [Alt+T] to toggle the toolpath off.
2. Choose
   - **Finish**
   - **Flowline**
   - **All**
   - **Surfaces**
   - **Done**
3. Right-click the tool display area and select **Get tool from library**.
4. Select the ½" ball endmill and **OK**.
5. Enter the values shown on the following dialog box.



6. Select the **Surface parameters** tab.
7. Enter the values shown on the following dialog box.



8. Select the **Finish flowline parameters** tab.
9. Enter the values shown on the following dialog box.

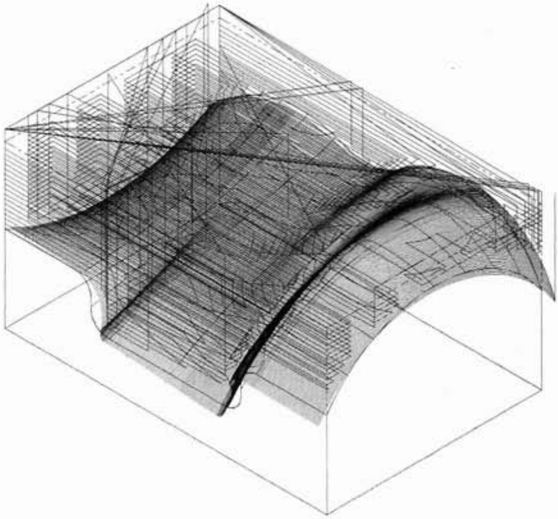


10. Choose **OK**.
11. Toggle **Offset** by clicking on it until the arrows are above the geometry surface.
12. Choose **Done**.

▶ *Backplot the toolpath*

1. Choose
   - **BACKUP**
   - **Operations**
   - **Select All**
   - **Backplot**
   - **Run**

The completed toolpath should look like the following picture.

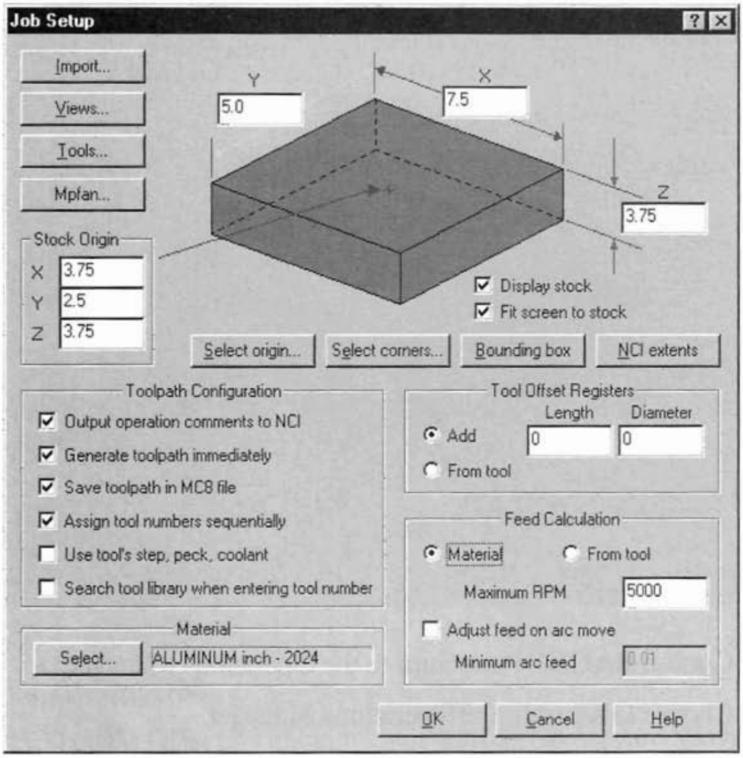


2. Choose **BACKUP** to return to the Operations Manager.
3. Choose **OK** to exit the Operations Manager.

Toolpath verification

▶ *Define the stock*

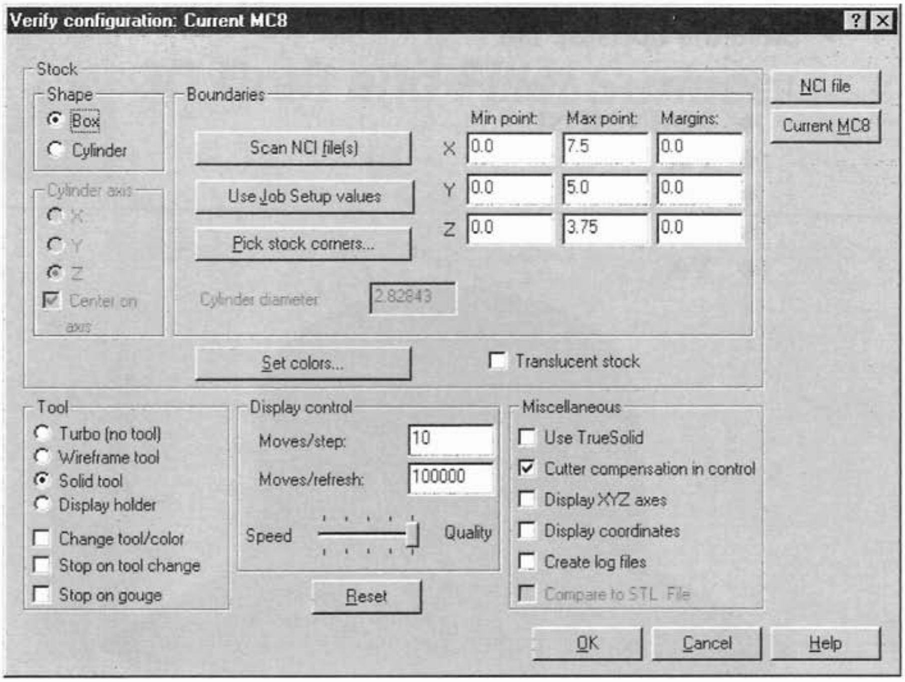
1. Choose **Job setup**.
2. Enter the values shown on the following dialog box.



3. Choose **OK**.

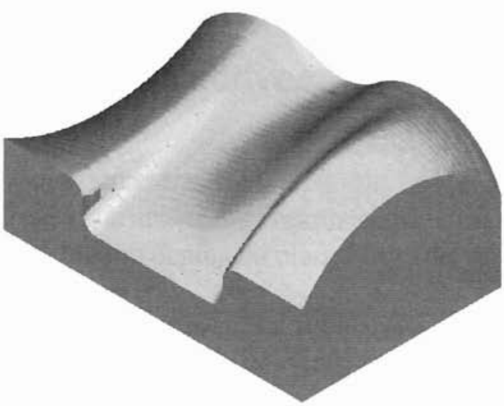
▶ *Verify the toolpath*

1. Choose **Operations**.
2. Choose
   - **Select All**
   - **Verify**
3. Choose the **Configure** button.
4. Select **Use Job Setup** values and enter the values shown in the following dialog box.



5. Choose **OK**.
6. Choose the **Machine** button.

Your part should look like the following picture.

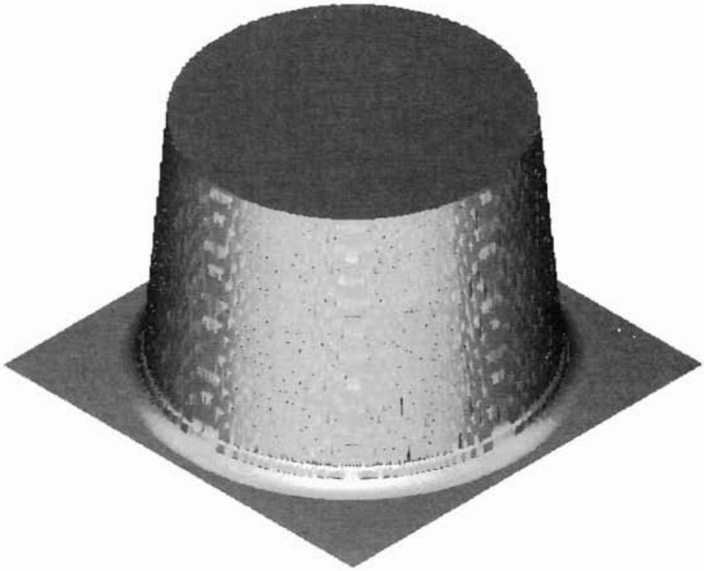


7. Exit the Verify toolbar.
8. Choose **OK** to exit the Operations Manager.

Save the updated file

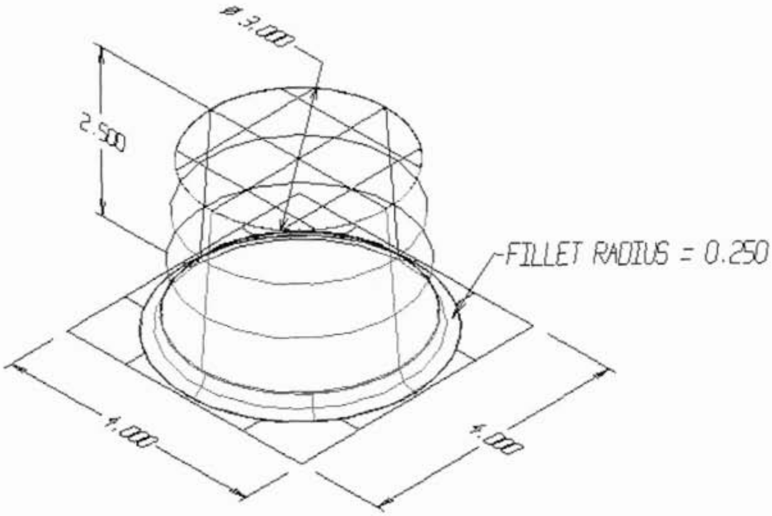
1. Choose
   - **MAIN MENU**
   - **File**
   - **Save**
   - **Save**
   - **Yes**

17 3D Draft and Fillet Surface



Objectives

- Design a 3D wireframe drawing.
- Create rectangles and arcs using coordinate positioning.
- Create the construction depth and plane of the drawing entities.
- Create a draft surface.
- Create a flat boundary surface.
- Create a fillet surface.
- Create a 3D toolpath.
- Perform solid model verification of the toolpath.

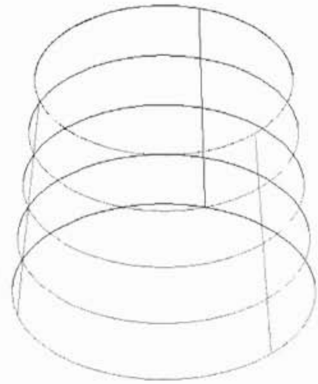


Geometry creation

▶ *Create the draft surface*

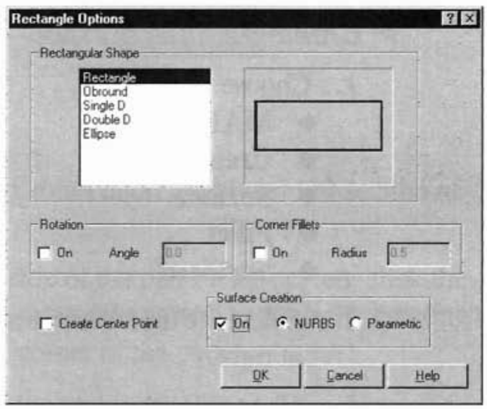
1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Arc**
   - ◆ **Circ pt+dia**
2. *Enter the diameter.* **3**
3. Choose **Origin**.
4. Press [Esc].
5. Change the Gview to **Isometric**.
6. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Surface**
   - ◆ **Draft**

7. Select the circle.
8. Choose
   - **Done**
   - **Length**
9. *Enter the length.* **–3**
10. Choose **Angle**.
11. *Enter draft angle.* **–5**
12. Choose **Do it**. The part should look like the picture at right.

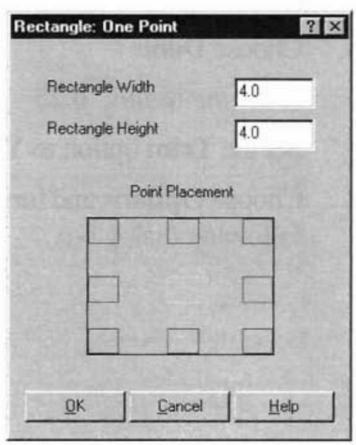


▶ *Create the ruled surface*

1. Choose the **Fit** button on the toolbar.
2. Change the current Z depth to **–2.5**.
3. Choose
   - **MAIN MENU**
   - **Create**
   - **Rectangle**
   - **Options**
4. Choose **On** in the **Surface Creation** section of the dialog box as shown at right.
5. Choose **OK**.
6. Choose **1 point**.

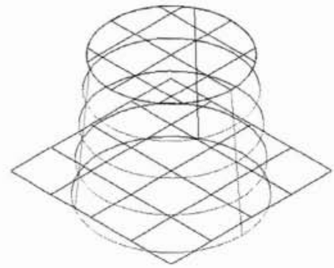


7. Enter the values shown in the dialog box at right.
8. Choose **OK**.
9. Choose **Origin**.



▶ *Create the flat boundary surface*

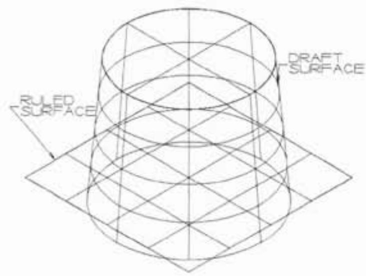
1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Surface**
   - ◆ **Trim/extend**
   - ◆ **Flat bndy**
   - ◆ **Chain**
2. Select the original (top) circle.
3. Choose
   - ◆ **Done**
   - ◆ **Do it**



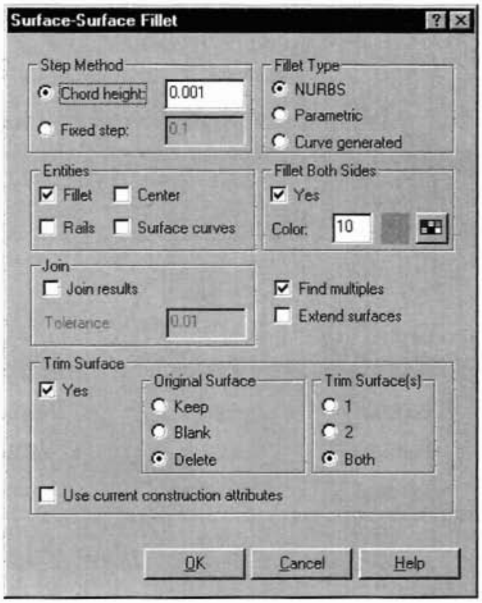
Your part should look like the one at right.

▶ *Create the fillet surface*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Create**
   - ◆ **Surface**
   - ◆ **Fillet**
   - ◆ **Surf/surf**
2. Select the draft surface as shown at right.

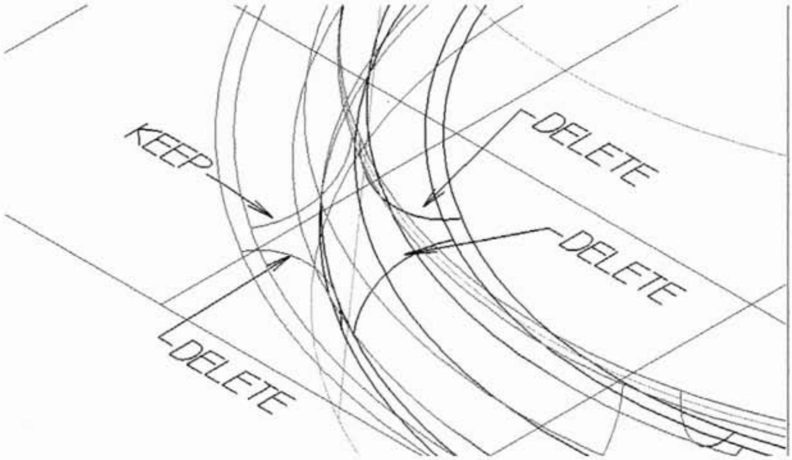


3. Choose **Done**.
4. Select the ruled surface.
5. Choose **Done**.
6. *Enter the radius.* **0.25**
7. Set the **Trim** option to **Y**.
8. Choose **Options** and turn on **Fillet both sides** as shown in the following dialog box.



9. Choose **OK**.
10. Choose **Do it**.

11. Select the **Screen-zoom** icon to zoom in on the lower- left section of the part.
12. Starting halfway down the left side of the part on the screen, click the left mouse button, and drag the mouse down and to the right, creating a window around the bottom left corner of the part.
13. Click once more to zoom in on the windowed area.
14. Select the 3 fillets (arcs) you wish to delete as shown in the following picture.



15. Select the **Screen-unzoom** icon from the toolbar.

16. Choose **BACKUP**.

Save the file

1. Choose
   - **MAIN MENU**
   - **File**
   - **Save**

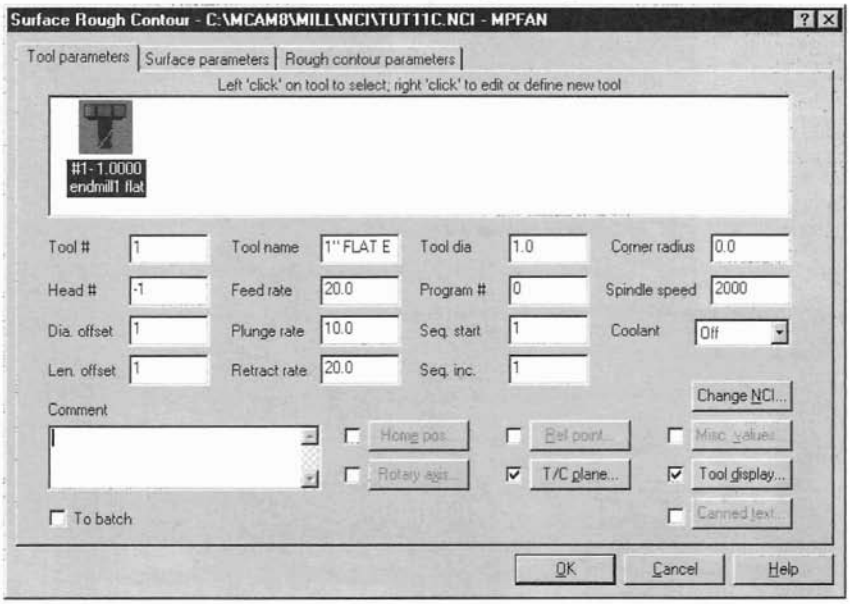
2. Save using your name and the number 17.

Toolpath creation

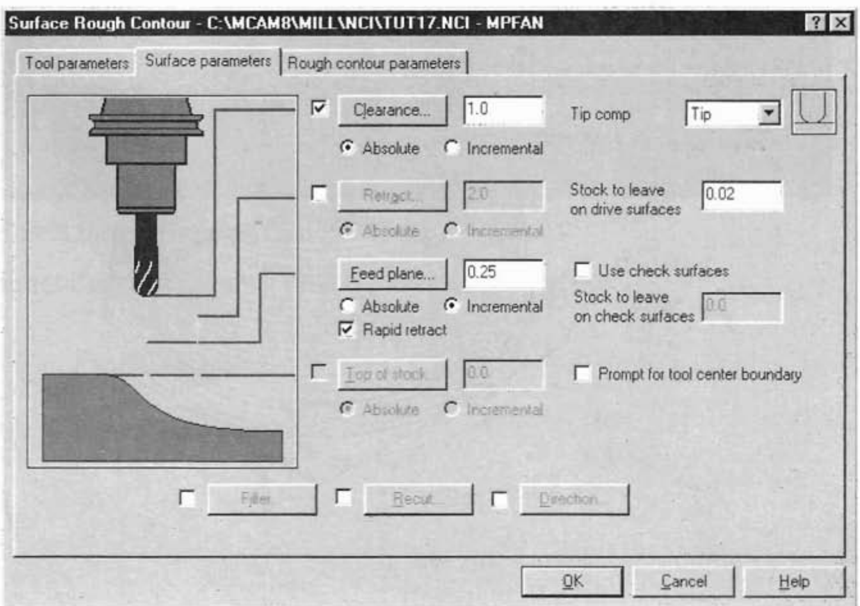
▶ *Rough out the surface*

1. Choose
   - **MAIN MENU**
   - **Toolpaths**
   - **Surface**
   - **Rough**
   - **Contour**
   - **All**
   - **Surfaces**
   - **Done**

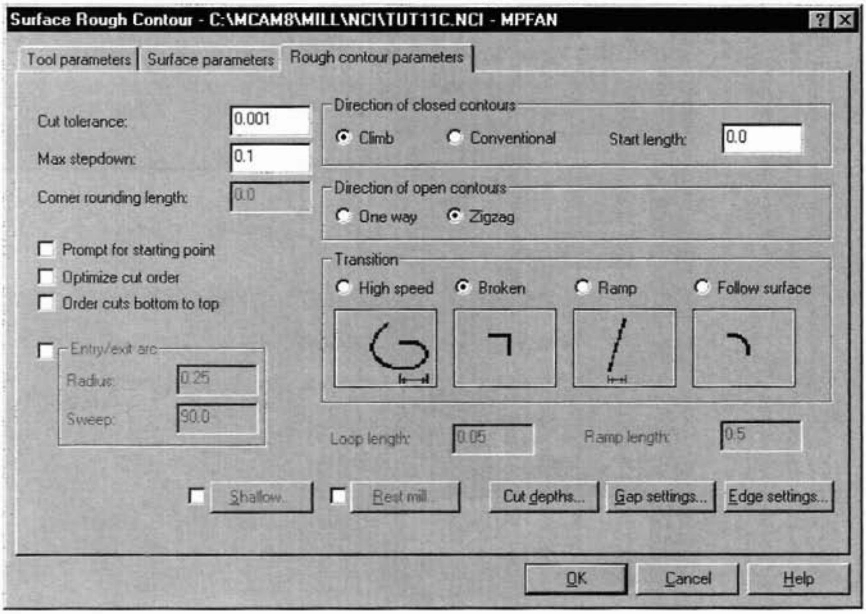
2. Right-click in the tool display window, select **Get tool from library,** and choose the 1" flat endmill.
3. Choose **OK**.
4. Enter the values shown on the following dialog box.



5. Select the **Surface parameters** tab.
6. Enter the values shown on the following dialog box.

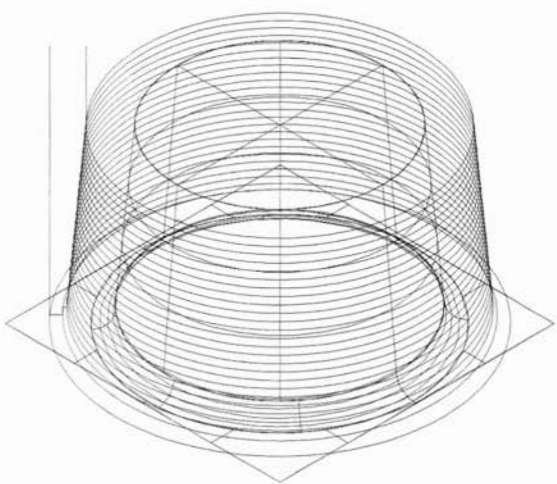


7. Select **Rough contour parameters** tab.
8. Enter the values shown on the following dialog box.



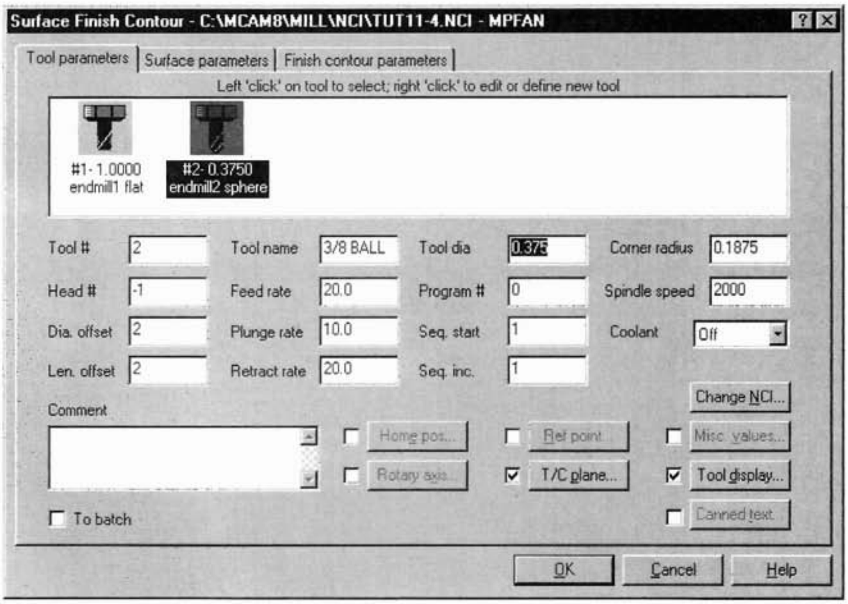
9. Choose **OK**.

The toolpath should look like the following picture.

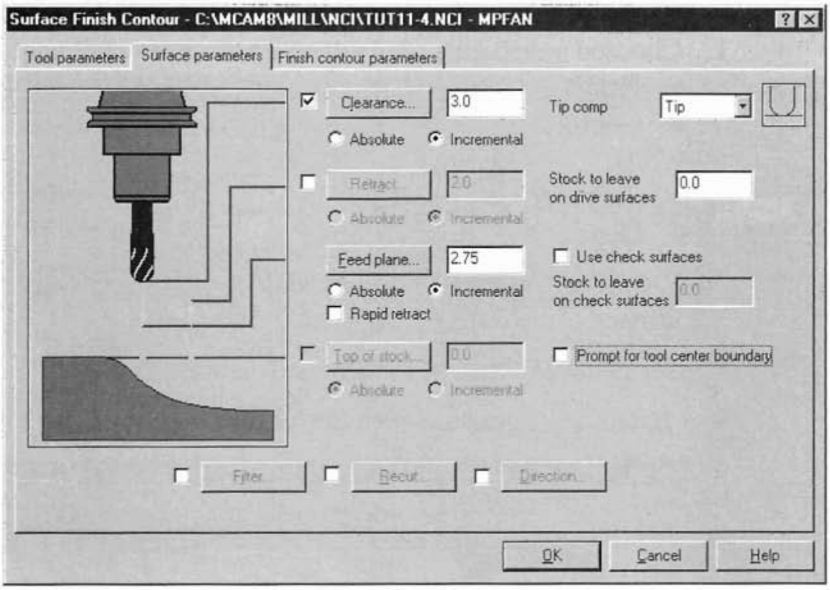


▶ *Finish the part*

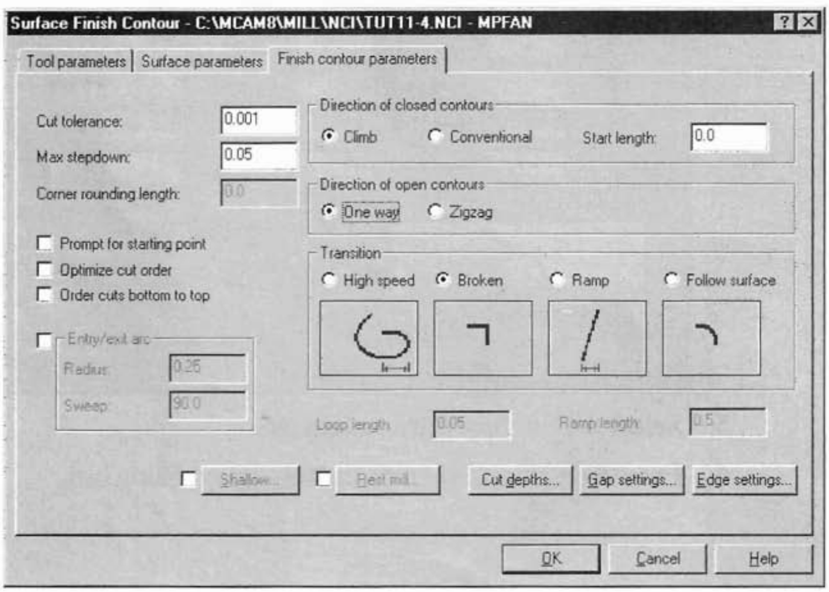
1. Choose
   - **Finish**
   - **Contour**
   - **All**
   - **Surfaces**
   - **Done**
2. Right-click on the tool display window and select **Get tool from library**.
3. Select the 3/8" endmill sphere and choose **OK**.
4. Enter the values shown on the following dialog box.



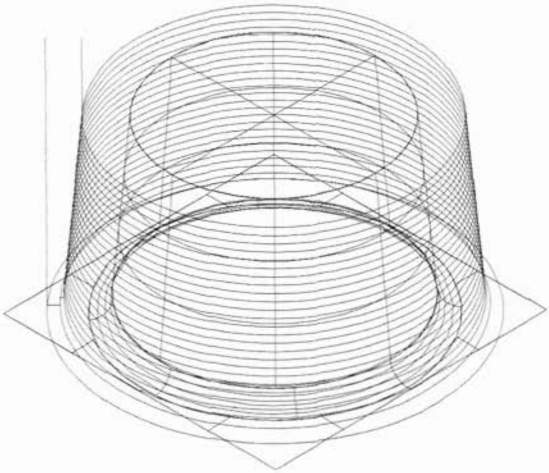
5. Select the **Surface parameters** tab.
6. Enter the values shown on the following dialog box.



7. Select the **Finish contour parameters** tab.



8. Choose **OK**. The completed toolpath should look like the following picture.

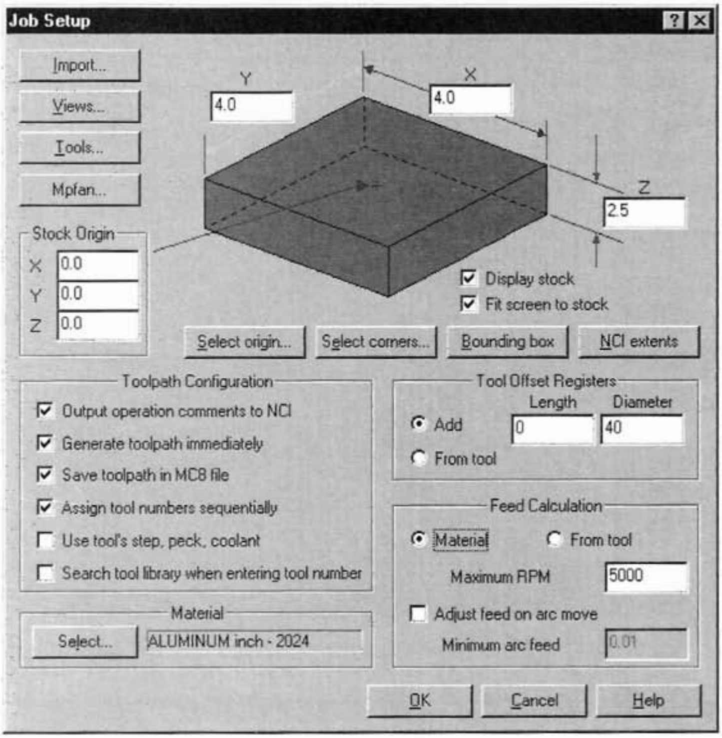


▶ *Backplot the toolpath*

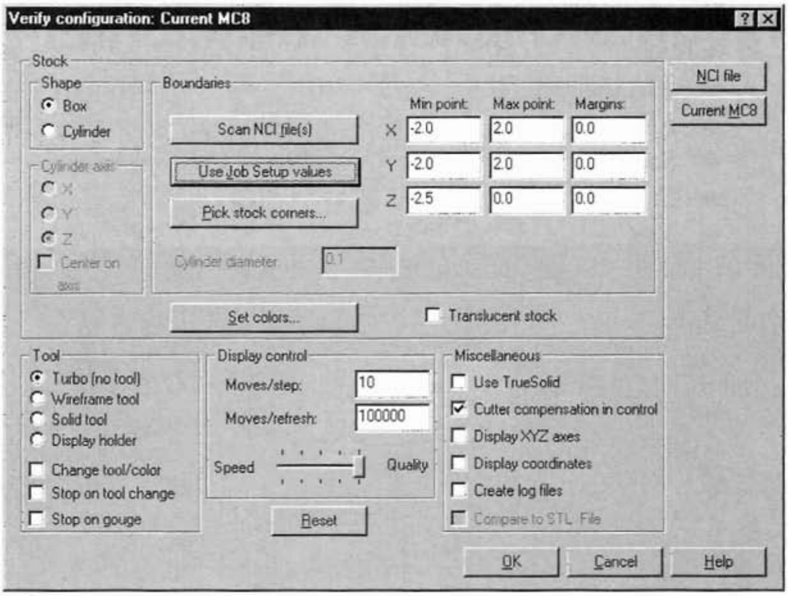
1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Toolpaths**
   - ◆ **Operations**
   - ◆ **Select All**
   - ◆ **Backplot**
   - ◆ **Run**
2. Choose **BACKUP** to return to the Operations Manager.
3. Choose **OK** to exit the Operations Manager.

▶ *Verify the toolpath*

1. Choose
   - ◆ **MAIN MENU**
   - ◆ **Toolpaths**
   - ◆ **Job setup**
2. Enter the values shown.

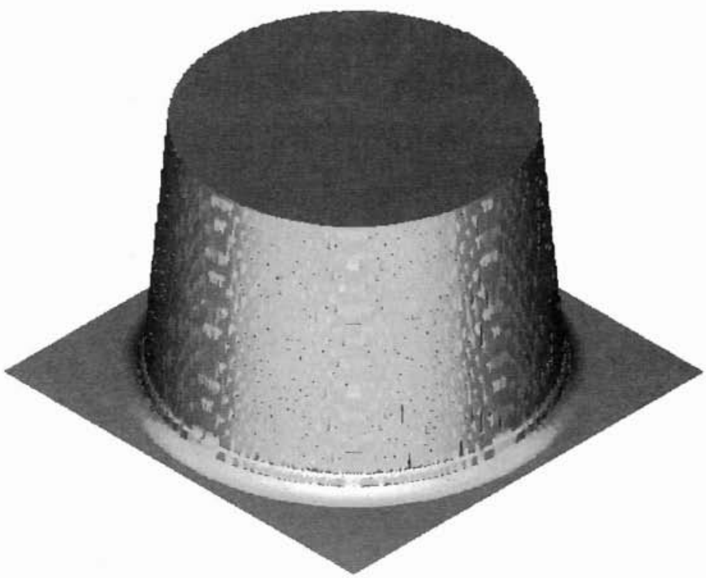


3. Select **OK**.
4. Choose **Operations**.
5. Select the **Verify** button.
6. Choose **Configure** from the Verify toolbar.
7. Select the **Use Job Setup values** button and enter the values shown on the following dialog box.



8. Choose **OK**.
9. Choose the **Machine** button.

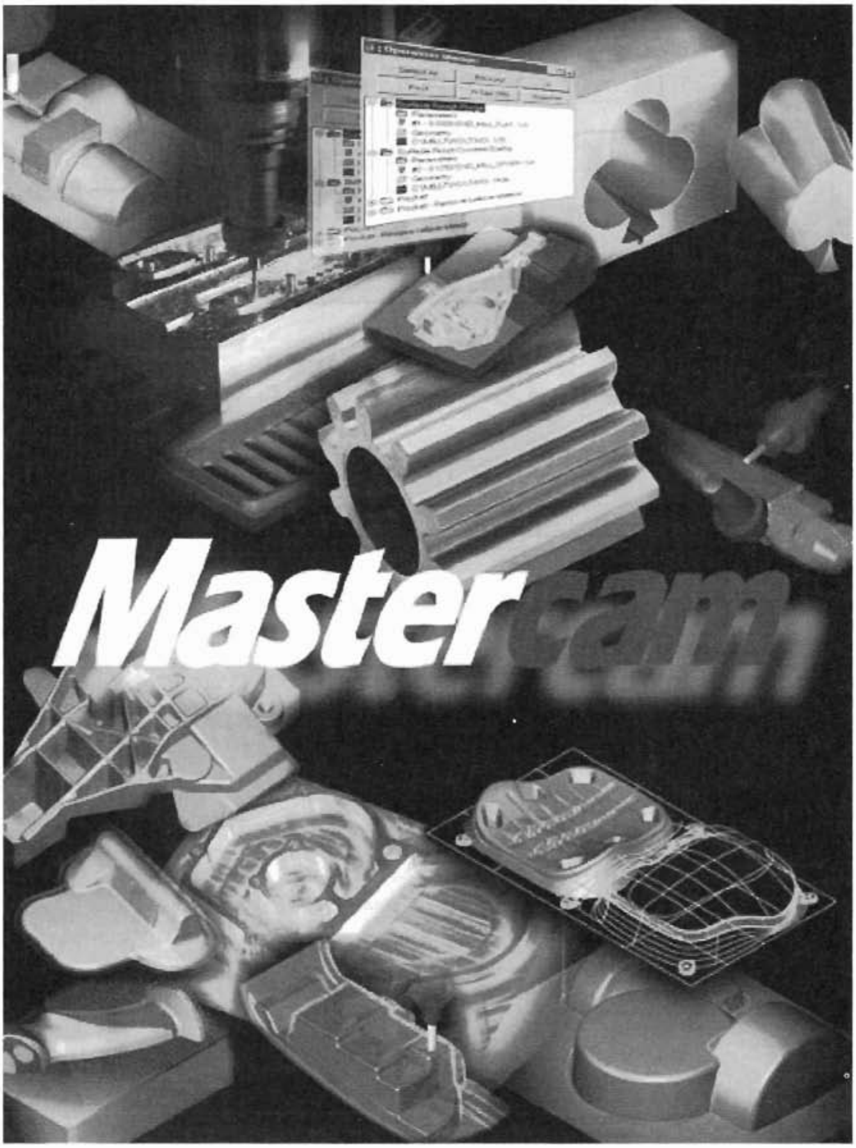
The part should look like the following picture.



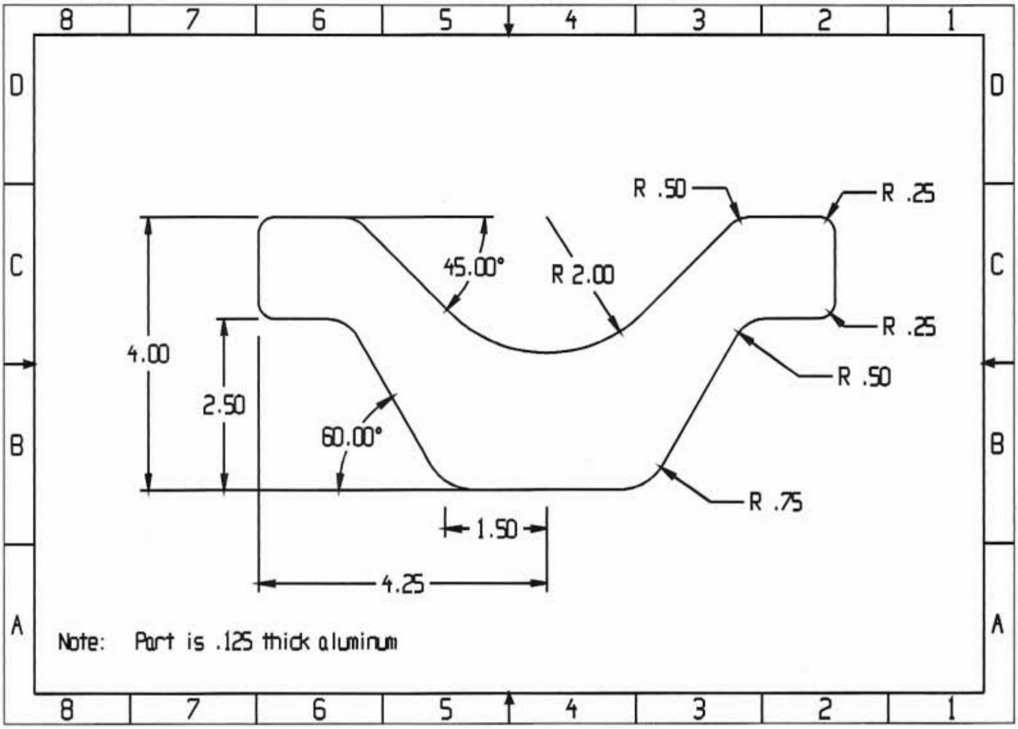
▶ *Save the file*

1. Choose
   - **MAIN MENU**
   - **File**
   - **Save**
   - **Save**
   - **Yes**

18 Enrichment Tutorial Drawings

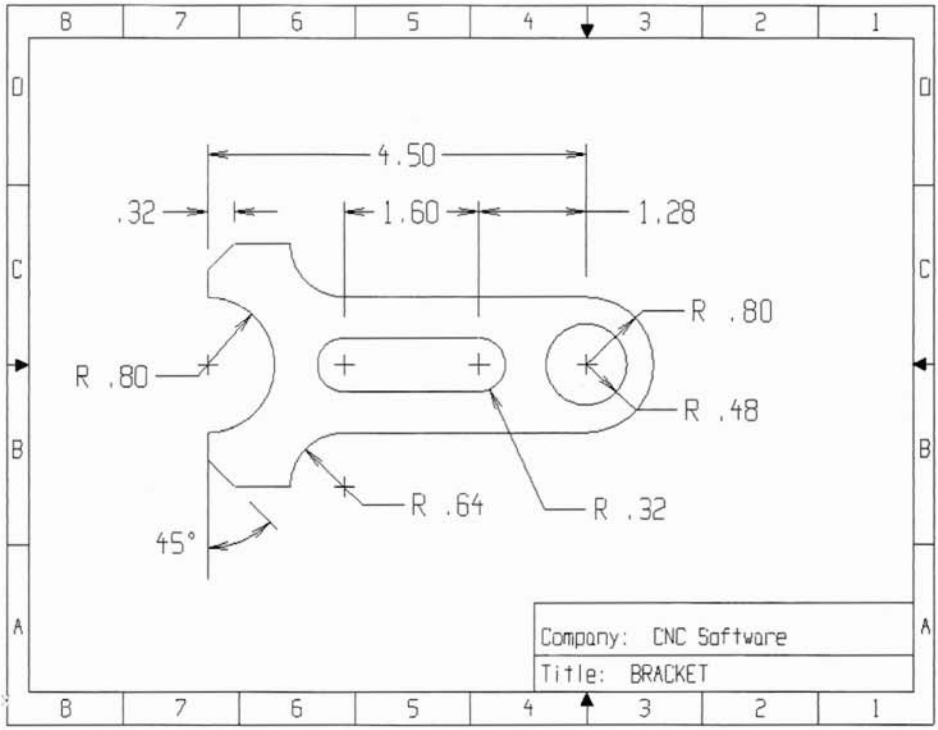


Exercise 1



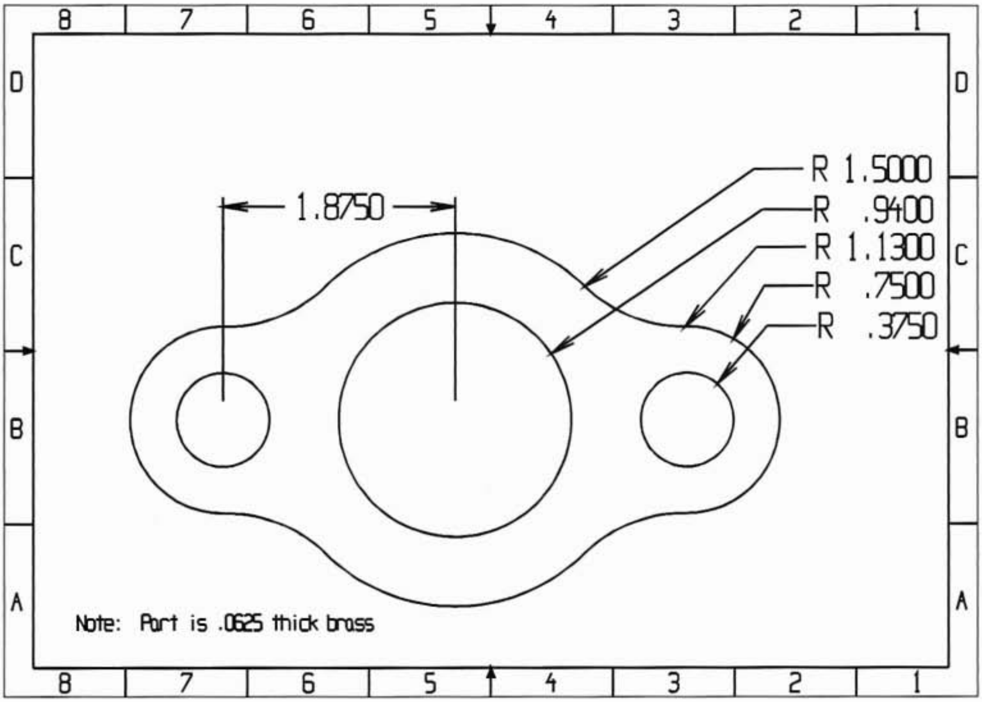
Make a part on your own from a print.

Exercise 2



Make a part from a print.

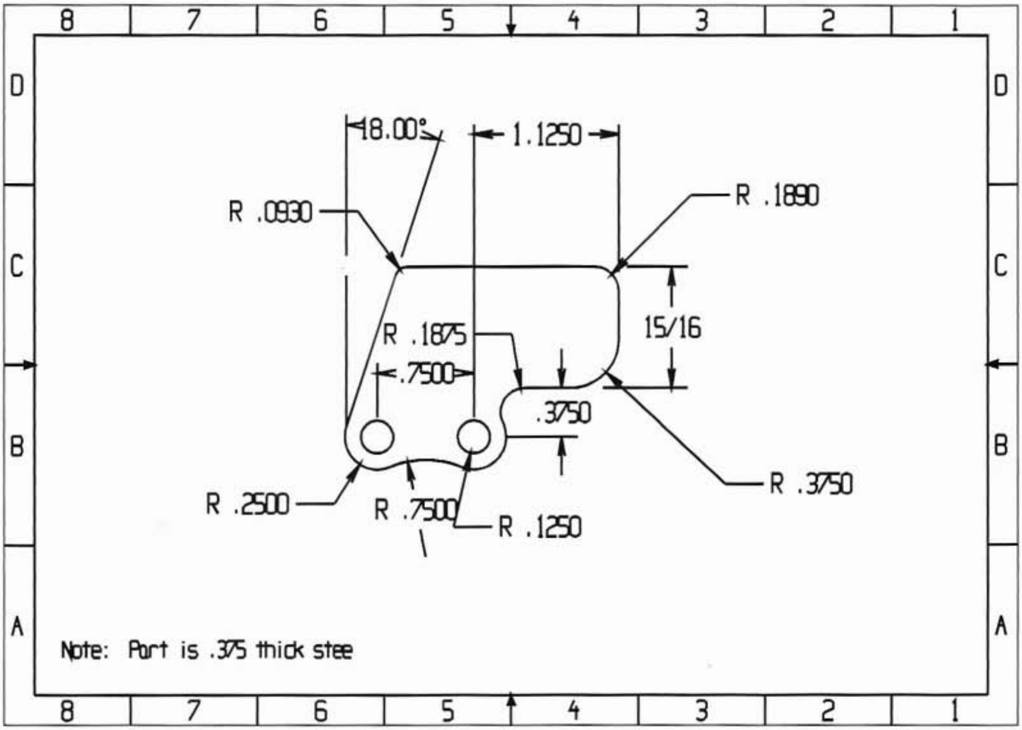
Exercise 3



Make a part from a print. Pay special attention to trimming on this part.

Tip: A circle has one break point at zero.

Exercise 4



Make a part from a drawing. This part can be particularly challenging. Remember to ask for help when you need it.

Glossary

2 ½D (contour)	A toolpath consisting of multiple sections in which the depth can vary between sections but is constant within a section.
2D (contour or plane)	A toolpath or geometry that lies in a single plane.
3D (contour, plane or space)	A toolpath or geometry defined in X, Y, and Z axes simultaneously; consists of lines, arc, parametric splines, and NURBS splines.
4-axis	Toolpaths defined by X, Y, and Z locations, but with a tool axis with an additional degree of freedom, permitting the tool to be oriented parallel to an axis other than X, Y, or Z.
5-axis	Toolpaths defined by X, Y, and Z locations, but with a tool axis with two additional degrees of freedom, permitting the tool to be oriented parallel to an axis other than X, Y, or Z.

A

A axis	Axis of circular motion about the X axis; expressed in degrees.
absolute (coordinates, dimensioning, positioning)	Measured from a fixed reference point, usually 0,0,0.
along entity	A series of evenly spaced points along a line, arc, or spline.
arc	An open or closed planar curve in which all positions are at a fixed distance (radius) from the center of the curve. A circle is a 360-degree arc.

associativity (toolpath, dimensioning, and solids)	A relationship that links geometry with toolpath, tool, material, and parameter information to create a complete toolpath operation. Permits modifications to geometry or machining parameters to easily regenerate accurate, updated solid topology, dimensions and toolpaths. Also the relationship between dimensioning and its geometry.
attribute data	Attributes of entities: level, color, style, width.
AutoCursor	A feature that snaps the cursor to endpoints, midpoints, intersections, center points, quadrants or an arc, and the origin points in the vicinity of the cursor; automates and speeds point detection.
AutoHighlight	A feature that speeds and simplifies entity selection by dynamically highlighting the entity under the cursor before then entity is actually selected.
AutoSave	Feature that automatically saves current geometry and operations at a regular time interval.

B

B axis	Axis of circular motion about the Y axis; expressed in degrees.
backplot	A feature that displays the path a tool takes to cut a part.
BBS	Acronym for Bulletin Board Service.
bitmap	A graphic composed of small dots that form shapes and curves; bitmap files use the BMP extension.
blank	To reduce the complexity of the graphics window by temporarily making one or more entities invisible. They remain blanked until the user selects and unblanks them. The blanked entities remain in the database and are saved with the file. See also **hide**.
blend	Smooth connection of surfaces.
bolt circle	Circular array of evenly spaced points defined by the center, radius, and a number of points on the circle.
boss	In general, a plateau of material from a surrounding cavity.

boundary	An edge, border, or limit; a curve or chain that indicates an edge.
bounding box	A feature in Job Setup used to approximate the limits of the stock required to machine a part. Also in Design.
branch (point)	Point in a chain where the endpoints of three or more entities meet.
browse	In Mastercam, to scan actual images of geometry files (MC8, MC7, or GE3) in a selected directory. Also to scan the file names in a directory.
b-spline	Basis spline, a representation of a curve as a piecewise collection of polynomials formed by a polygon; may be rational or non-rational See also **NURBS spline**.

C

C axis	Axis of circular motion about the Z axis; expressed in degrees.
CAD	Acronym for computer-aided design.
CAD/CAM	Acronym for a combined CAD and CAM system.
CAM	Acronym for computer-aided manufacturing.
canned text	Post processor variables that can be associated with special commands, for example, an auto stop to check on a part during machining.
cartesian	Coordinate system using X, Y, and Z values to locate a point in space.
CFG	Mastercam configuration file extension.
chain	Selection of one or more curves (lines, arcs, and/or splines) that have adjoining endpoints and often form boundaries; may be open or closed. Point entities can be chained using the point method for tool rapid moves; curves and points can be chained.
chain direction	The order of curve selection in a chain from start point to endpoint in an open chain; in a closed chain, may be clockwise or counterclockwise.

chain synchronization (Sync mode)	To break a chain into separate sections, each beginning and ending at a specified point, then match it with one or more other chains with the same number of synchronization points.
chaining tolerance	Maximum distance between two endpoints that can still be chained.
chamfer	Beveled or sloping edge that consists of one line that trims two intersecting lines. Each endpoint of the chamfer is positioned at a defined distance from the intersection of the two selected lines. In contour toolpaths, a chamfer is used to break sharp edges.
check surface	A surface or solid face that the system protects during toolpath generation on another surface.
C-Hook	Custom-made Mastercam application program created in the C or C++ programming language. C-Hooks that are automatically installed with Mastercam and appear on menus with an asterisk (*) after the name.
chord height (tolerance)	In general, the amount of play allowed between a surface edge and the original geometry; determines the degree of precision with which edges of trimmed surfaces are created. See also **edge tolerance**. Also the tolerance with which Mastercam calculates surface shading independent of current display scale.
circle	A closed planar curve in which all positions are at a fixed distance (radius) from the center of the curve.
circle mill	A function that generates a toolpath to automatically machine full circles with an entry arc, two 180-degree arcs, and an exit arc.
clearance plane or height	Height at which the tool moves between two separate machining operations.
climb milling	Cutting in which the tool rotates in a direction opposite the direction of travel along the side being cut. Generally produces a smoother surface finish than conventional milling. When the spindle is rotating clockwise, climb milling may be achieved by setting cutter compensation to the left. See also **conventional milling**.

closed chain	A chain whose start and end points are identical.
CNC	Acronym for computer numerical control, which is a computer used to control machine tools.
CNCEDIT	File editor supplied with Mastercam that also provides some CNC and DNC capabilities.
collinear	Having the property of lying on the same line.
combine view	Combines all parallel views into a single view and moves arcs from separate parallel views to a single view.
communications (serial)	Transmission of information, one bit at a time over a single line, between a PC and any devices attached to it. See also **communications parameters**.
communications parameters	Parameters that control the transfer of information between a PC and devices attached to it. Parameters include format, port, baud rate, parity, data bits, stop bits, echo terminal emulation, strip carriage returns, strip line feeds, EOL (end of line) delay, and DOS communications mode. Communications is a File menu option (Communic).
compensation in computer	Offset in the toolpath that compensates for the radius of the cutting tool; made in the computer.
composite curve	A chain of curves that meet endpoint to endpoint.
construction origin	Reference point (X0, Y0, Z0) for geometry creation; the same as the system origin unless reassigned by the user.
construction plane (Cplane)	Plane where geometry is created; may be different from the graphics view (Gview). Mastercam provides several standard construction planes: 3D, top, front, back, bottom, left and right side, isometric, and axonometric. Additional planes can be created.
context-sensitive help	Helpful information displayed on the screen that is relevant to the operation being performed.
contour	Path described by two or more axes. Also a method of analyzing selected boundaries or the boundary offset, thus simulating toolpath creation.

control points	Points that define a NURBS spline; usually do not lie on the spline.
conventional milling	Cutting in which the tool rotates in the same direction as the direction of travel along the side being cut. Selecting clockwise spindle rotation and cutter compensation to the right results in conventional milling. See also **climb milling**.
converter	A function that imports or exports geometry files in formats other than Mastercam and translates them to or from Mastercam format. Formats that can be translated include ASCII, CADL, DWG, DXF, IGES, NFL, Parasolid, SAT, STEP, STL, VDA, GEO, old GE3, and pre-version 7 materials, tools and parameter libraries.
Coons patch	A surface constructed by blending a grid of along curves and across curves. Named after Steven A. Coons. See also **Coons surface**.
Coons surface	A surface composed of one or more Coons patches.
copious data	An entity type that represents a collection of geometric forms (points and lines). Copious data originates in an IGES file. Mastercam can convert it to points and lines during translation. The Modify, Break, Cdata/line function can also be used to convert copious data to points and lines.
critical depths	Toolpath cut depths that must be machined even if depth increments must be adjusted to cut them.
cross-section	A section made by a plane cutting traversely through solids or surfaces. Also used in project toolpaths.
curvature (surface)	Measure of curving of a curve or surface.
curve	Line, arc, spline, or surface curve.
cut (toolpaths and solids)	When used with respect to toolpaths, refers to tool movement in the Z axis; do not confuse with pass. When used with respect to solids, a type of solid operation in which chains of curves are extruded, revolved, swept, or lofted as material is removed from an existing solid (target body).

cutter compensation	Compensation for the radius of the cutting tool. In contour analysis and toolpath generation, determines which direction the system offsets the selected boundary with respect to chain direction and tool radius. See also **compensation in computer** and **compensation in control**.
cutter offset	Distance from the part surface to the axial tool center; tool radius.
cutter path	The path the center or tip of the tool follows over the part.

D

data bits	A communications parameter that defines the number of bits used to represent a character; must be the same for both the PC and the CNC controller or peripheral device.
depth cuts	Z-axis cuts that the tool makes in a contour, pocket, face, circle mill, or surface toolpath to get to the final depth in set increments.
DF8	Mastercam default parameter file format for Version 8 (*.DF8); contains default values for all toolpath types.
dirty operation	A solid or toolpath operation that has been modified in some way; for example, its parameters or geometry input. The system marks dirty solids and operations with a red 'X' in the Solids Manager dialog box or Operations Manager. When an operation is dirty, it must be regenerated for the toolpath or geometry to match the parameters.
display cues	Features that clarify how geometry is oriented in the graphics window: XYZ axes marker, dynamic arrow, surface backside display.
display list	An internal feature that saves the display data for each entity; used by Mastercam to determine what entities are visible on the screen and to speed redraws, view changes, and other screen functions.

DNC	Acronym for direct numerical control or distributive numerical control. Direct numerical control uses a single computer to simultaneously control operation of a group of NC machines. Distributive numerical control uses a network of computers to coordinate operation of a group of CNC machine tools. Mastercam can be used in either situation.
dongle	Another name for a SIM, which is required to run Mastercam.
DOS (shell)	Acronym for Disk Operating System. A DOS shell can be used execute MS-DOS commands while Mastercam is running.
double D	A shape composed of two line entities and two arc entities.
dpi	Dots per inch, a measure of graphic resolution.
drafting entity	An entity used in dimensioning: witness lines, leader lines, dimensions, cross hatches, labels, notes, copious data.
drive surface	A surface and/or solid body that undergoes a surface or multiaxis machining operation. See also **check surface**.
dynamic arrow	Cursor display that permits dynamic movement along geometry to indicate a position; changes size to indicate orientation of arrow relative to viewer. When large, the arrow points toward viewer. When small, the arrow points away.

E

edge	A topological element of a solid model, which has an underlying curve.
edge profile	Defines the shape of the surface outer boundaries.
edge tolerance	The degree of precision with which edges of trimmed surfaces are created.
editor	An application used to modify files of certain types. See also **MCEDIT, PFEDIT32, CNCEDIT**.
ellipse	An oval-shaped NURBS spline or collection of connected lines.

entity	A design building block. There are geometric entities (points, lines, arcs, splines, surface curves, surfaces, solids, copious data) and drafting entities (witness, lines, leader lines, dimensions, crosshatches, labels, notes).
entity association	The dependent relationship between one entity and a second entity or group of entities from which the first entity is generated.

F

feed plane	Height that the tool moves to before changing from the rapid rate to the plunge rate to enter the part.
feed rate	Cutting tool speed of movement in the cutting direction; usually expressed in inches per minute.
file information	Displayed when an operator presses [F9]: file name and path, date and time of last file save, file size in bytes, current display scale, relative positions of construction, tool, and system origins and axes.
fillet	An arc tangent to two non-tangent curves; a rounded interior pr exterior corner.
filter (Filter)	The process of eliminating unnecessary tool movements from a toolpath. Do not confuse with mask. When capitalized, a utility that performs this function.
finish	Precision surface machining.
fit screen	To display the visible geometry so as to fill as much of the graphics window as possible; a Mastercam function that is available from the right-click menu, from the toolbar, and by pressing [Alt + F1].
flat boundary	Used to create a flat, trimmed surface from one or more closed sets of curves.
flowline	Multiple curves along an entire surface in one constant parameter direction, that is, one of the directions in which the system creates the surface.

font	Text style. Mastercam fonts include Stick, Roman, European, Swiss, Hartford, Old English, Palatino, and Dayville. Windows® TrueType® fonts are also supported.
FPT	Feed per tooth.
free-form surface	A surface generated from arbitrarily shaped lines and curves; includes ruled, lofted, 2D swept, 3D swept, and Coons surfaces.
function	A single operation, for example, Analyze, Set Norms.
function keys	Keyboard keys numbered [F1] through [F10]; may be assigned to functions, C-Hooks, and macros.

G

Gcode	In general, an NC part program; specifically, a code that, among other things, defines part program coordinates.
GE3	Mastercam file format for geometry files prior to version 7 (*.GE3); does not contain toolpath information.
geometric entity	Points, lines, arcs, splines, surface curves, surfaces, solids.
geometric surface	Surface composed of constant geometric shapes: sphere, cones, cylinders, draft surfaces, and surfaces of revolution.
geometry	Data that defines the spatial placement and shape of the boundaries and surfaces of a geometric model (part).
global parameters	Dimension attributes that are applied to all drafting entities; includes dimension symbols, coordinate formats, tolerances, text properties, witness and leader line attributes.
gouge	The result or act of a tool machining away material that should not have been removed.
graphics view (Gview)	The point of view of the displayed geometry; may be top, front, side, isometric, as well as defined dynamically by the operator.
graphics window	Workspace area in Mastercam where the geometry displays.
group	A collection of entities or operations that can be manipulated as a single entity. See also **result**.
GUI	Acronym for graphic user interface.

H

hardcopy	Paper copy of the geometry visible in the graphics window.
HASP	Acronym for Hardware Against Software Piracy; refers to the type of SIM used by Mastercam 7.0 or later.
hide	To make all entities except those selected temporarily invisible so as to simplify the graphics window. They remain invisible until unhidden as a group. Hidden entities are not saved with the file. See also **blank**.
highlight	To select with the cursor, with the result that the selected object changes color or reverses to white type on a dark background. See also **AutoHighlight**.
home position	Position where the tool returns for tool changes and at the end of the NC program.
HSS	High speed steel.

I

icon	Small symbol used to simplify access to a program or function; sometimes also called a button.
IGES	Acronym for Initial Graphics Exchange Standard, an international neutral format; used to transfer geometry from one brand of CAD system to another.
incremental (coordinates, dimensioning, positioning)	Measured from the immediately preceding point.
infinite look ahead	In contour analysis, to search the entire boundary to find self-intersections based on the current offset distance and cutter compensation.
integer	A whole number such as 3, 50, or 764; used as a data type for counting or numbering.

J

job	Contains a set of operations.

Job Setup	Machining job parameters, including stock setup, NCI configuration, and tool offsets.
jump height command	Allows a tool to be moved to a height above the clearance plane between points in a toolpath.

L

level	A grouping used to organize geometry in Mastercam.
level report	A report of what entities exist on each level of a geometry file.
line	Straight entity between two endpoints.
line style	The appearance of a line; may be solid, hidden, center, phantom, or Zbreak.
linear array	A repeating toolpath along the X or Y axis of the construction plane at a specific distance.
linearization tolerance	Used when converting 3D arcs and 2D or 3D splines in the chained geometry from curves to lines; represents the maximum distance between an arc or spline and its linear approximation.
loft surface	A surface composed of smoothly blended curves created by fitting through a set of cross-sectional curves.

M

macro	Group of commands and instructions that can be stored, recalled, and executed to perform a task; may be used to automate common or repetitive tasks.
Main Menu	Presents primary Mastercam functions: Analyze, Create, File, Modify, Xform, Delete, Screen, Exit, and in Mill and Lathe, Toolpaths, and NC Utilities.
mask	Restricts entity selection to certain types or levels. Do not confuse with filter.
Mastercam®	An integrated CAD/CAM software package created by CNC Software, Inc.
material library	Contains information on materials for machining that is used to set a base percentage for feed rates and spindle speeds; uses the MT7 or MT8 file extension.

MC7	Format for a Mastercam file in Version 7 (*.MC7); contains a set of operations, geometry, toolpath parameters, material definition, NCI data, and tool information.
MC8	Format for a Mastercam file in Version 8 (*.MC8); contains geometry, toolpath parameters, material definition, NCI data, and tool information. See also **job** and **operation**.
MCEDIT	A Mastercam text editor; provides NC capabilities, file editing, and file manipulation capabilities. See also **PFE32** and **CNCEDIT**.
merge	To combine MC7, MC8, or GE3 files with the current geometry file. Some or all of one or more configuration files can also be combined.
MT8	Mastercam material library file format for Version 8.
MTL	Mastercam tool library file format for versions prior to Version 7 (*.MTL).
multiaxis	Using more than one axis; often refers to 4- or 5-axis toolpaths.

N

NC	Acronym for numerical control, a technique for controlling machine tools or processes by coded command instructions; also the file format output from Mastercam post processors.
NCI	Acronym for numerical control intermediate, the Mastercam intermediate toolpath file format.
node (spline)	Points in a parametric spline.
nonlinear	Not located on a single line.
normal (arrow)	Perpendicular to. There are two normal vectors for each planar chain of curves, which point in opposite directions. A normal arrow indicates the side of the selected surface on which the system creates the surface.
NURBS (spline)	Acronym for non-uniform rational b-spline; a two- or three-dimensional curve defined by knots and control points.

NURBS surface	A surface that is defined analogously to NURBS splines with the string of control points expanded in another direction resulting in a grid.

O

obround	A shape composed of two straight line entities and two 180-degree arc entities.
offset	To displace an entity or chain by a distance in a perpendicular direction relative to the current construction plane. In a curve, displacement is perpendicular to the direction vector at every location on the curve.
offset surface	A surface created by offsetting an existing surface by a distance.
OP8	Mastercam operation library file format for Version 8.
open chain	A chain whose first and last endpoints are not identical, such as a line.
OpenGL®	An operating system-independent standard for displaying graphics.
operation (toolpaths and solids)	When used with respect to toolpaths, consists of geometry, toolpath (NCI file), tool definition, material definition, and parameters. A set of operations makes up a job or MC8 file. Each operation includes only one toolpath. See also **job** and **MC8**. When used with respect to solids, the action or actions performed to create or modify a solid. Each operation, such as fillet or extrude, is listed separately in the history tree under the solid that it defines or modifies.
operation library	Contains default parameters for a specific toolpath; can be applied to current geometry; uses the OP7 or OP8 file extension.
Operations Manager	Lists all operations in the current MC8 file, including both associative and non-associative toolpaths, and offers options for managing them.

origin	Intersection point of coordinate axes. See also **system origin**, **construction origin**, and **tool origin**.

P

pan	To change the position of geometry in the graphics window by stepping through the area in a horizontal right or left direction, or vertical up or down direction using the arrow keys.
parallel views	Construction planes that exist in the same 2D plane but differ by rotation or position.
parametric spline	A 2D or 3D curve defined by a set of coefficients or nodes; mathematically equivalent to non-rational Bézier splines.
parametric surface	A surface composed of parametric splines in which each curve segment is expanded in another direction resulting in a patch.
part	The item to be machined.
part drawing	Describes the shape and size of a part; usually includes part features, dimensions, tolerances, and surface roughness.
part feature	The distinctive shape and size to be produced in a part; can be 2D (flat surfaces, internal and external profiles, pockets, holes, etc.) or 3D (surfaces).
pass	A tool movement in the X and Y axes. Do not confuse with cut.
patch	Area of a surface bounded by four segments of the generating curves.
peck	A tool move that occurs at the programmed feed rate as it feeds into and retracts out of the stock during a drill toolpath.
peck clearance	Depth that the tool rapids down to between peck movements during a drill toolpath.
PFE32	A Mastercam text editor; provides file editing and manipulation capabilities.
planar	Flat, lying within a single geometric plane.
plot	To output current graphics window to a plotter or file.

point (entity)	Entity that marks a position in 2D or 3D space but that has no dimension.
point (using the mouse)	To move the mouse until the mouse pointer on the screen rests on the item you want.
point data	Data consisting only of points.
polar (coordinates and dimension)	Coordinate system that uses a known point, length (radius), and angle to locate a point in space. The angle is calculated in a counterclockwise direction from the positive horizontal axis that runs through the known point in the current construction plane.
polygon	Irregular, closed shape with three or more straight sides. In Mastercam, can be created as a single NURBS spline or as a collection of individual lines.
port	A physical connection on a PC. Serial ports are used to connect to the CNC controller and are identified as COM1, COM2, etc.
post	Post processor. Also a post processor (PST) file.
post processor	A program that translates NCI data to a format usable by a machine, that is, to an NC part program or Gcode.
primitive	A surface or solid created using a predefined shape, such as a block or sphere. The parameters can be changed interactively, but it maintains its original shape. A primitive surface or solid is not defined by curve geometry. Mastercam primitives include cylinder, cone, block, extrusion (surfaces only), sphere, and torus.
PRM	Mastercam default parameter file format and file extension for versions prior to Version 7.
prompt area	A two- or four-line area at the bottom of the Mastercam interface used to display data or enter values with the keyboard.
PST	A post processor file and extension.

Q

quadrant	A section of a plane in which quadrant 1 lies between 0 and 90 degrees, quadrant 2 lies between 90 and 180 degrees, quadrant 3 lies between 180 and 270 degrees, and quadrant 4 lies between 270 and 360 degrees.

R

RAM	Acronym for random-access memory.
RAM-saver	An option that compacts the system database and frees up available RAM; can also perform an efficiency and integrity check on the database.
real number	A number that can be represented by digits in a numbering system with a fixed base, such as 0.5 or 25.4; used for storing measurements and other values to some limit of precision.
rectangle	Parallelogram composed of four straight lines and four right angles.
redraw	To erase then redisplay visible geometry in the graphics window to clean up display remnants.
reference point	Point to which the tool moves before reentering a toolpath.
regenerate	In general, to recompute solids, drafting entities, or toolpaths when associated geometry or parameters have been modified. To rebuild the graphics window display list so as to improve the speed and results. The Regen path option in the Operations Manager recomputes a toolpath when the associated geometry or parameters have been modified.
relative (coordinates, dimensioning, and positioning)	Distance measured from specific point, not necessarily the zero or preceding point.
repaint	To erase then redisplay the visible geometry in the graphics window to clean up display remnants.
required pilot diameter	Minimum diameter necessary for the tool to enter the toolpath.

result	The appearance of an entity group that has been transformed; may be selected for further transformation or translation. The default color of a result is purple.
retract amount	Distance that the drill retracts every time it makes a peck move during a drill toolpath.
retract height	The height to which the tool moves before the next tool pass.
revolved surface	A surface created by rotating a sectional shape around an axis or line.
right-click	To click on something using the right mouse button; displays alternate (right-click) menus.
right-click menu	A menu that opens when you right-click the mouse; gives quick access to many common features.
roll	To wrap a line, arc, or spline around a cylinder.
rough	To remove large amounts of material as rapidly as possible.
RPM	Revolutions per minute; a measure of spindle speed.
rubber-band	Temporary display of entities that will be created or modified; the display updates dynamically based on the cursor location to indicate the result with the cursor at that location.
ruled surface	A surface composed of linearly blended curves created by connecting straight lines between two or more lines or curves.

S

save some	To save selected entities to an MC8 file. Toolpaths cannot be saved using this method.
scale	To increase or decrease the size of an entity by a factor relative to the construction origin or some other point. Also see scaleXYZ.
scaleXYZ	To increase or decrease the size of an entity independently in X, Y, and Z dimensions. Also see scale.
Screen, Configure	A menu that sets Mastercam's default values. Default configuration files are MILL8.CFG (English units) and MILL8M.CFG (metric units).

segment	A section of a spline between two nodes
selection cues	In Mastercam, a way of gathering data from the graphics window; also called shortcuts. Allows you to modify data collected from the graphics window by entering values in the prompt area. Shortcuts appear in the prompt area as X, Y ,Z, R(adius), D(iameter), L(ine length), S(distance between two points), and A(ngle).
selection grid	A grid of reference points that the cursor can snap to during sketching.
setup sheet	An ASCII file created by Mastercam that contains NCI file information including operation, tool reference, total programming time, and text entered manually during programming; uses the SET extension.
SFM	Acronym for surface feet per minute.
shading	Representation of light striking a colored surface or solid object using gradated fill.
shortcuts	In Mastercam, a way of gathering data from the graphics window; also called selection cues. Allows you to modify data collected from the graphics window by entering values in the prompt area. Shortcuts appear in the prompt area as X, Y, Z, R(adius), D(iameter), L(ine length), S(distance between two points), and A(ngle).
SIM	Acronym for Software Interface Module; sometimes called a dongle; required to run Mastercam.
single D	A shape composed of one line entity and one arc entity.
sketch	To create geometry or select entities by identifying points in the graphics window using the cursor and mouse.
slice	The process of creating points at the intersection of lines, arcs, and splines with a plane and creating points where they intersect. Also the process of creating curves at the intersection of surfaces and solids with a plane and creating curves where they intersect.

solid	A geometric representation of a closed three-dimensional object. In Mastercam, a solid is a geometric entity that differs from other types of geometric entities such as lines, arcs, splines in that each solid is also a topological entity that occupies a region of space and that consists of one or more faces, which define the closed boundary of the solid.
spindle speed	Tool rotation speed (RPM)
spline	Smooth, free-form curve controlled by points including the condition of its endpoints; may be parametric or NURBS spline.
startup file	Configuration file, which contains Mastercam default values.
statistics (screen)	Tally of visible entities by type.
stepdown	The distance that separates adjacent cuts in the Z axis on a surface toolpath.
stepover	The distance that separates adjacent cuts in the XY plane on a surface toolpath.
stretch	To place around geometry a window that intersects other geometry, then to translate the entities that are completely inside the window and also lengthen or shorten any lines that cross the window (by translating the endpoint that is inside the window).
style/width	Line style and width used to display lines, arcs, and splines.
subprogram	A section of the NCI file that repeats at different locations.
supplementary angle	An angle that when added to another angle produces an angle of 180 degrees.
surface	A representation of a part's skin by mathematical equations; a boundary defining an exterior face of a solid model.
surface curve	A curve entity type created directly on a surface through the Create Curve function.
surface memory allocation	The amount of RAM allocated for surface generation.
surface model	Defines a surface, including the edges of each surface.

solid	A geometric representation of a closed three-dimensional object. In Mastercam, a solid is a geometric entity that differs from other types of geometric entities such as lines, arcs, splines in that each solid is also a topological entity that occupies a region of space and that consists of one or more faces, which define the closed boundary of the solid.
spindle speed	Tool rotation speed (RPM)
spline	Smooth, free-form curve controlled by points including the condition of its endpoints; may be parametric or NURBS spline.
startup file	Configuration file, which contains Mastercam default values.
statistics (screen)	Tally of visible entities by type.
stepdown	The distance that separates adjacent cuts in the Z axis on a surface toolpath.
stepover	The distance that separates adjacent cuts in the XY plane on a surface toolpath.
stretch	To place around geometry a window that intersects other geometry, then to translate the entities that are completely inside the window and also lengthen or shorten any lines that cross the window (by translating the endpoint that is inside the window).
style/width	Line style and width used to display lines, arcs, and splines.
subprogram	A section of the NCI file that repeats at different locations.
supplementary angle	An angle that when added to another angle produces an angle of 180 degrees.
surface	A representation of a part's skin by mathematical equations; a boundary defining an exterior face of a solid model.
surface curve	A curve entity type created directly on a surface through the Create Curve function.
surface memory allocation	The amount of RAM allocated for surface generation.
surface model	Defines a surface, including the edges of each surface.

surface normal	Vector perpendicular to tangent plane of surface.
surface projection	Creates points (or curves) by projecting selected points (or curves) onto selected surfaces.
surface shading	Color fill added to surfaces and solids to make them more easily visible; may be full-time or studio.
surface types	Mastercam supports three surface types based on mathematical generation methods: parametric, NURBS, and curve-generated. Surfaces may also be typed by components and application into loft, ruled, Coons, revolved, swept, draft, fillet, offset, trim/extend, and blend surfaces.
swept surface	Created by sweeping one or two curves or chains of curves (across curves) through a trajectory of one or two other curves or chains of curves (along curves); may be 2D or 3D. Also called a drag surface.
Sync	A function that breaks a chain into separate sections, each beginning and ending at a specified point, then matches it with one or more other chains with the same number of synchronization points.
system origin	Fixed reference point for all geometry creation (X0, Y0, Z0).
system tolerance	Maximum distance between two points that can still be considered coincident.

T

tangent	Two curves whose slope is continuous in direction across their common endpoint.
tip comp	Cutter compensation calculated to the tool center or tip.
TL8	Mastercam tool library file format for Version 8.
tolerance	The precision with which an entity must fit another entity or process, or the maximum permissible deviation from a value; includes system, chaining, minimum arc length, curve minimum step size, curve maximum step size, curve chordal deviation, and maximum surface deviation tolerances. Tolerance dimension format is one of the global drafting parameters.

tool	The cutting or machining part, usually removable, of a lathe, planer, drill, or similar machine.
tool body	The body or bodies that are added to, removed from, or used to keep a common region with a selected target body during a Boolean operation. Once a solid is designated a tool body, it becomes part of the target body. In the Solids Manager dialog box, a tool body is listed under the solid and Boolean operation that it helps to define, and its icon is marked with the letter 'T'. Note: When you delete a Boolean operation, the system restores the operation's tool bodies as distinct, active solids. You can also duplicate a tool body to obtain an active copy of the solid.
tool center boundary	A closed set of curves that limits tool movement for a surface toolpath. The tool's center stays within the selected boundary.
tool library	Contains information on multiple mill and lathe tools, such as spindle speeds, plunge rates, and tool diameters; uses the TL7 or TL8 file extension.
Tools Manager	A Mastercam function that provides a list of tools stored in the current job or in the current tool library; also allows management of tool libraries.
tool origin	The reference point (X0, Y0, Z0) in the tool plane (Tplane); the same as the system origin unless reassigned by the user.
tool plane (Tplane)	A 2D plane that represents the CNC machine's XY axis and origin; also called Tplane.
toolbar	Area on the screen that contains icons (buttons). The buttons are arranged in pages to which the user can scroll; may be moved and reassigned.
toolpath	Shows where a tool removes material from a part.
Tplane	Abbreviation for tool plane; a 2D plane that represents the CNC machine's XY axis and origin.
transform	To translate, mirror, rotate, scale, offset stretch, or roll geometry or toolpaths.

translate	To move or copy geometry or toolpaths to a new location without changing orientation. Also see transform.
trim	To act as a boundary for a entity or surface.
trim/extend surface	A surface created by trimming or extending existing surfaces.
trimmed surface	Surface bounded at a set of edges; can be created by applying any or a number of processes to untrimmed or trimmed surfaces, for example, projection of curves, intersection, or filleting with other surfaces.

U

undo	To reverse the last action performed.
unwrap	To unroll a rolled entity.
unzoom	To return to the previous display scale or to the original display size.
unzoom by 0.8	To return to the previous display scale or reduce the size of the displayed geometry to 80% of its original size.

V

vector	A directed line segment.
vertex	An endpoint of an edge.
view	Angle of observing the geometry – top, front, back, bottom, right side, left side, Cplane, isometric, or axonometric.
viewport	Area within the graphics window that displays the geometry.

W

window (selection)	A polygon sketched around entities to select them.
wireframe model	Three-dimensional object composed of separate lines joined to create a model; a complete set of edge and skin profiles that create a surface.
witness (dimension) lines	Thin solid lines that project from a dimensioned object to indicate the extent of the leader lines.
work offset	A value that shifts the origin and coordinate system of the tool plane when creating toolpaths at different locations (for example, tombstone work).

X

X axis	Horizontal axis relative to the construction origin; right of origin is positive; left of origin is negative. See also **Cplane**.
Xform	Abbreviation for transform, a function that can translate, mirror, rotate, scale, offset, stretch, and roll geometry.
XYZ axes marker	Indicates the axis orientation according to 3D space; displayed in the bottom left corner of the graphics window; updates to reflect the current graphics view (Gview).

Y

Y axis	Vertical axis relative to the construction origin; above origin is positive; below origin is negative. See also **Cplane**.

Z

Z axis	Perpendicular to the X and Y axis relative to the construction origin. See also **Cplane**.
Z depth	Current construction depth, which is the depth of the currently defined construction plane (Cplane) relative to the system origin.
zoom	To magnify a rectangular portion of the graphics window.

Shortcut Keys

Alt + 0	Set Z depth for Cplane
Alt + 1	Set main color
Alt + 2	Set main level
Alt + 3	Set mask level
Alt + 4	Set tool plane (menu)
Alt + 5	Set Cplane (menu)
Alt + 6	Set Gview (menu)
Alt + A	AutoSave
Alt + B	Toolbar on/off
Alt + C	Run C-Hooks
Alt + D	Drafting global parameters
Alt + E	Hide/unhide geometry
Alt + F	Menu font
Alt + G	Selection grid parameters
Alt + H	On-line help
Alt + I	List open files
Alt + J	Job setup
Alt + L	Set line style and width
Alt + M	Memory allocations
Alt + N	Edit named views
Alt + O	Operations Manager
Alt + P	Prompt area on/off
Alt + Q	Undo last operation
Alt + R	Edit last operation
Alt + S	Full-time shading on/off
Alt + T	In Toolpath menu, turn toolpath display on/off
Alt + U	Undo last action

Alt + V	Mastercam version number and SIM serial number
Alt + W	Viewport configuration
Alt + X	Set main color, level, line style and width from selected entity
Alt + Z	Set visible levels
Alt + '	Create two-point circle
Alt + Tab	Switch between applications
Alt + -	With hidden entities, select additional entities to hide
Alt + =	Unhide selected entities
Alt + F1	Fit geometry to screen
Alt + F2	Unzoom by 0.8
Alt + F3	Cursor tracking on/off
Alt + F4	Exit Mastercam
Alt + F5	Delete using window selection
Alt + F7	Blank geometry
Alt + F8	System configuration
Alt + F9	Display all axes
F1	Zoom
F2	Unzoom
F3	Repaint
F4	Analyze menu
F5	Delete menu
F6	File menu
F7	Modify menu

F8	Create menu
F9	Part information on/off
F10	List all functions and execute selected
Tab / Shift + Tab	Navigate between controls in dialog boxes
Esc	System interrupt or menu backup
Page up	Zoom in by 0.8
Page down	Zoom out by 0.8
Arrow keys	Pan
Ctrl + E	In Operations Manager, expand or collapse all operations

General Notes

Menu Commands

When the system first starts, the Mastercam logo screen appears, and then goes to the geometry screen where the Main Menu is displayed. It includes:
Analyze: Displays all relevant information about a point, line, arc, spline, surface or dimension on the screen.
Create: Adds geometry to the system's database and draws it on the screen.
File: Manipulates disk files (saves, retrieves, converts, transmits, receives, etc.).
Modify: Alters screen geometry with the fillet, trim, break, and join commands.
Xform: Transforms geometry with the mirror, rotate, scale, translate, and offset commands.
Delete: Removes an entity or group of entities from the screen and the system's database.
Screen: Plots the geometry currently displayed on the screen, changes the current color or level, displays entity endpoints or changes the way in which geometry is displayed.
Solids: Creates and modifies solid geometry.

Toolpaths: Programs NC toolpaths (CAM systems only).
NC utils: Manipulates NC toolpaths (CAM systems only).

The Secondary Menu

Z: Use Z depth to change the current construction depth. The construction depth is the depth of the currently defined construction plane. This value may be entered by selecting an existing point (i.e., endpoint of a line) or by typing [Z] and the desired depth, followed by [Enter]. The construction depth is changed automatically in most creation functions to the 3D-construction mode.
Color: Assigns a system color. This instructs the system to display any new geometry created in the selected color (Create,Line; Create,Arc; Backplot,Save geometry; etc.). It also applies the chosen color to a converted file that does not support colors (i.e., DXF). Mastercam contains two color palettes from which you can select system colors: a 16-color and 256-color palette.

Level: Assigns a system level to created/converted geometry. Level stores any new geometry created in the selected level. It also applies the chosen level to a converted file that does not support levels (i.e., NFL or ASCII). This feature also controls what levels are visible. You can set a total of 256 levels in Mastercam.
Style/Width: Sets the current line style and a width factor (between 1 and 5)
Mask: Sets the system's masking level. With the masking level OFF, the system will recognize any entity in the database; however, if the mask level is set (changed to a non-zero level number between 1 and 255) the system will then only recognize entities that are on that level. Mask will affect all functions (delete, group, file conversion, etc.) except fit, screen statistics, or screen endpoints. To reset the masking level to OFF, the masking level must be set to 0.

Tplane: The tool plane (Tplane) is a two dimensional plane that represents a CNC machine's coordinate system.
Cplane: Defines the plane on which you will create and manipulate geometry. You have several construction plane (Cplane) options.
Gview: Alters the way in which you view images through the graphics view (Gview) feature. Gview allows you to view geometry from several different angles.

Data Entry Shortcuts

Mastercam lets you use several shortcuts to enter information into text boxes when creating geometry. These shortcuts let you get data from an existing entity and use it when creating new geometry.
The shortcuts are as follows:

A Use "A" to input an existing angle.
D Use "D" to input an existing diameter.
L Use "L" to input an existing length.
R Use "R" to input an existing radius.
S Use "S" to input the distance between two existing points.
X Use "X" to input an existing X coordinate.
Y Use "Y" to input an existing Y coordinate.
Z Use "Z" to input an existing Z coordinate.

To use a shortcut:

Type a letter from the list above, instead of a value, into a text box.
Press [Enter]. Mastercam prompts you to select the existing entity from which you want to get the data.
Select the entity you want to use.

Mastercam displays the data it gets from the existing entity.
Press [Enter] to accept the data or type a different value and press [Enter].
Mastercam uses the data from the existing entity in creating the new entity.

Data Entry Shortcut Example:

This example uses the "L" data entry shortcut to create a line.

1. Create a line with endpoints at 0,0,0 and 3,0,0.
   This will be the existing line from which you get data by using "L."
2. Choose **MAIN MENU**, Create, Line, Polar.
3. Set 0,0,0 as one endpoint of the polar line.
4. Set 33 degrees as the angle of the line.
5. Type [L] and press [Enter] for the line length.

Mastercam prompts you to select an existing line.

6. Select the existing line.

Mastercam displays the data it gets from the existing line.

7. Press [Enter] to accept this data.

Mastercam creates the new line.

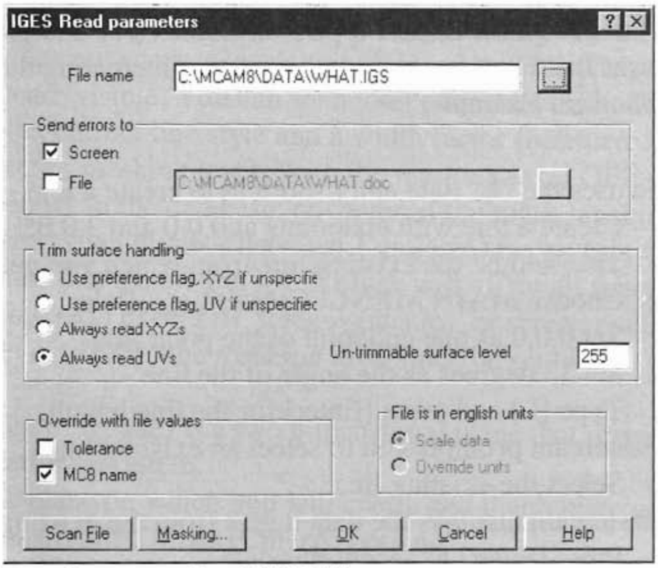
Graphics views

There are 8 standard graphics views in Mastercam.

View 1 =	Top
View 2 =	Front
View 3 =	Back
View 4 =	Bottom
View 5 =	Right
View 6 =	Left
View 7 =	Isometric
View 8 =	Axonometric

IGES translator

IGES is a public standard. It is used by many full-scale CAD systems. The IGES standard is considerably more complex than other file formats. The IGES interface is three-dimensional.



Use the Help file for assistance using IGES files.

DWG translator

The DWG format is used by AUTOCAD® (Autodesk, Inc.). The DWG interface is three-dimensional and supports colors and levels. Use Mastercam Help for more information on converting files.

DXF translator

The DXF (Drawing Interchange File) format is used by AUTOCAD® (Autodesk, Inc.) Splines are not supported (use IGES). Use Mastercam Help for more information on converting files.

Note: Use IGES conversion with AutoCAD® version 10 and later (not DXF).

C-Hooks

Mastercam C-Hooks are custom-made application programs created in the C programming language. Anyone, including CNC Software, Inc., Mastercam dealers, and our customers, can develop a C-Hook that runs with Mastercam. C-Hooks use the same graphic and menu functions as Mastercam.

Cutter Compensation

Cutter compensation gives you the ability to compensate for the radius of the cutting tool. You can use two methods to assign cutter compensation: in the controller or in the computer. Assigning cutter compensation in the controller sends a command for cutter compensation to the cutting system directly from the controller. Assigning cutter compensation in the computer allows you to generate a toolpath with the compensation.

Cutter Comp In Control

This parameter outputs a command for cutter compensation in the CNC system. You can select "Right," "Off," or "Left." Select "Right" and the system will place the cutter to the right of the part (G42); select "Left" and the system places the cutter to the left of the part (G41). Select "Off" and the system will ignore cutter compensation within the controller.

Cutter Comp In Computer

This parameter builds cutter compensation directly into the toolpath. You can select "Right," "Off," or "Left." Select "Right" and the system will place the cutter to the right of the part; select "Left" and the system places the cutter to the left of the part. Select "Off" and the system will ignore cutter compensation within the computer. This parameter also determines the internal offset direction when you program multiple cuts. If you set the Cutter Compensation - In Computer parameter to "Off," the system will determine the offset direction by the Cutter Compensation - In Control parameter. If both types of cutter compensation are set to "Off," this offset direction is unknown and all the roughing and finishing cuts will overlap. To avoid this condition, set the cutter compensation to left or right, and set the cutter diameter to 0. Setting this parameter to "Off" will also cancel the output of entry and exit arcs.

Compensate to Cutter - Center/Tip

This parameter allows you to set cutter compensation to the cutting tool's center or tip.

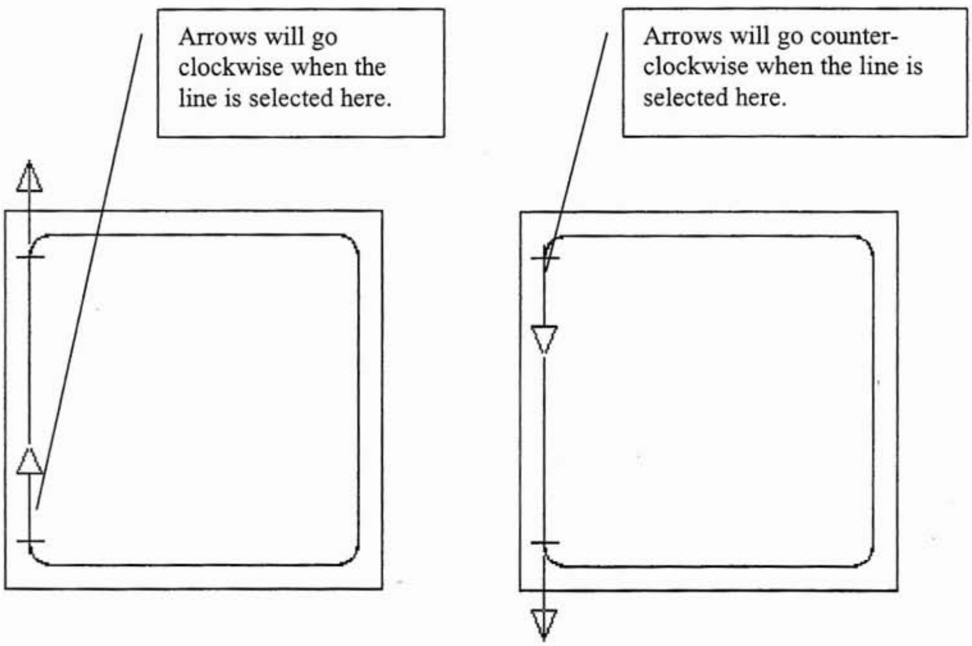
Chaining

Chaining is a process used to select a number of entities, (lines, arcs, points, splines etc.) Chaining is used when modifying or transforming existing entities or when creating a toolpath.

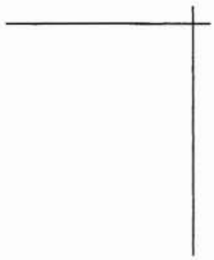
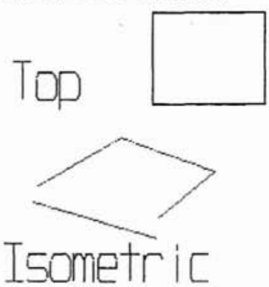
How to Chain

When chaining geometry to create a toolpath, it is important in certain circumstances to chain the geometry in the proper direction, since it will determine the direction the cutter travels when machining the part. The following pictures are examples of how to select geometry when chaining for a toolpath.

Note: To change the direction of the arrows, choose Reverse from the Mastercam chaining menu.

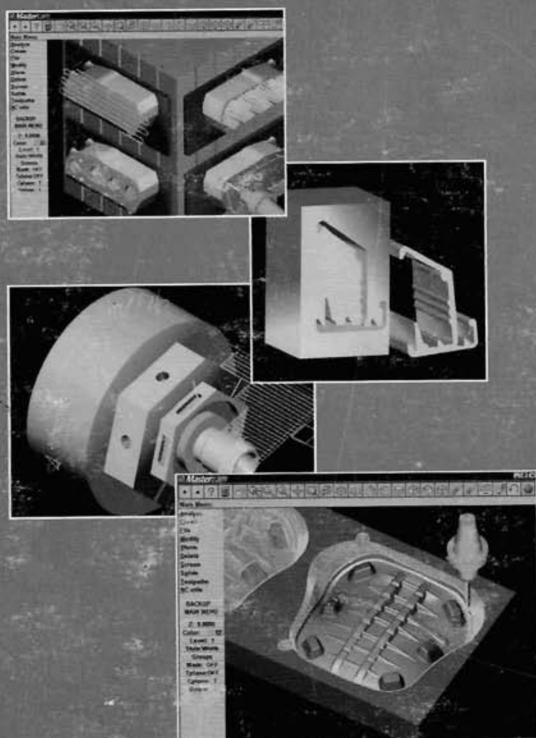


Chaining Problems Diagnostic

Chain stops. Lines cross. 	Zoom in to inspect.	Trim lines
Chain stops at branch points 	Unselect and select **Options** from the menu.	Choose **Plane mask** from the chaining options.
Lines don't touch 	Look at geometry in Gview Isometric. Unselect the entity and select **Options** from the menu.	Choose **Ignore depths** from the chaining options.
Chains erratically	Duplicate entities on top of each other.	**MAIN MENU Delete Duplicate Entities**
Chain stops in midline	Entities of differing lengths on top of each other.	Delete lines or entities and redraw them.

The Leader in PC-based CAD/CAM Software.

Mastercam offers CAD/CAM software for basic 2-axis machining, multi-axis milling and turning, wire EDM, 3D design, drafting, surface and solid modeling. Whatever your machining needs, there is a Mastercam product for your budget and application.



CNC software, inc.

Educational Division
Suite 2A • 5717 Wollochet Drive NW
Gig Harbor, WA 98335 • USA
http:/ /www.mastercam.com
mcinfo@mastercam.com

Printed on recycled paper

Learning Series Mill Projects & Apps ISBN: 1-88833